Numerical Aerodynamic Simulation Program

IIOC
TECHNICAL SUMMARIES

July 1986 - February 1987

(NASA-TM-102897) NUMERICAL AERODYNAMIC SIMULATION PROGRAM. IIOC TECHNICAL SUMMARIES Interim Report, Jul. 1986 - Feb. 1987 (NASA) 70 p

N90-70694

Unclas
00/02 0257171

#0257171

NUMERICAL AERODYNAMIC SIMULATION PROGRAM

IIOC

TECHNICAL

SUMMARIES

JULY 1986 - FEBRUARY 1987

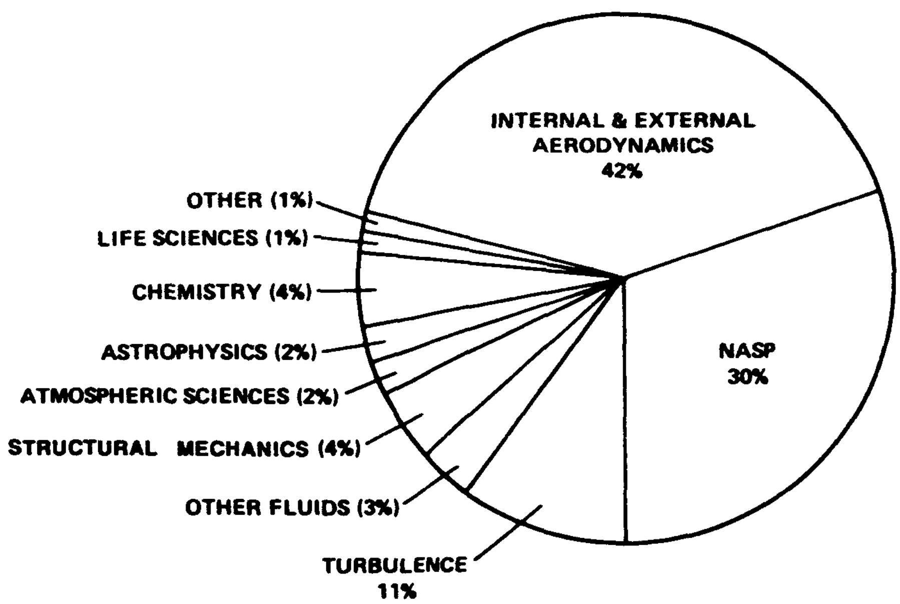
INTRODUCTION

This report contains selected scientific results for the IIOC (Interim Initial Operational Configuration) period of the Numerical Aerodynamic Simulation (NAS) Program. This interim period lasted from July 1986 through February 1987 and accommodated 240 scientists conducting research using the initial capabilities for the NAS Processing System Network (NPSN). It was during this period that the NPSN was undergoing rapid development in preparation for the first full year of operations which began in March 1987.

The organization that composed the initial set of scientists represented government, industry, and university communities from all over the United States. The breakdown of scientists by major usage categories was as follows:

NASA Ames Research Center	109
NASA Langley Research Center	53
NASA Lewis Research Center	11
Other NASA Centers	5
Industry	27
Department of Defense	30
Other Government Agencies	5

The research that was accomplished during the IIOC period was represented by the following scientific/program disciplines based on 123 projects:



At the completion of each operational year, it is expected that an annual publication will follow to provide a compilation of significant scientific results for that year. If you have comments about this annual report or want additional copies, please contact the NAS User Interface Manager at the following address:

NAS Systems Division
NASA Ames Research Center
Mail Stop 258-5
Moffett Field, CA 94035

TABLE OF CONTENTS

NAS IIOC PROJECTS

IIOC

TECHNICAL

SUMMARIES

NUMERICAL SOLUTION OF THE REYNOLDS-AVERAGED NAVIER-STOKES EQUATIONS FOR FLOW ABOUT AN ALMOST COMPLETE AIRCRAFT

Ramesh K. Agarwal, Principal Investigator
Co-investigator: Jerry E. Deese
McDonnell Douglas Research Laboratories

Research Objective:
To develop a computational code capable of predicting the viscous flow field over a complete aircraft.

Approach:
Slender-layer Reynolds-averaged Navier-Stokes equations are solved on a global body-conforming curvilinear grid by employing Jameson's finite-volume Runge-Kutta time-stepping scheme. The global grid about the aircraft is generated by employing a three-dimensional grid-generation code based on a hybrid mapping/elliptic partial differential equation method. Turbulence effects are modeled using a simple algebraic eddy-viscosity model. Variable time-step and implicit smoothing of the residuals are employed to enhance convergence to steady state.

Accomplishment Description:
The slender layer wing-body code MDSSL30 has been installed on the Cray-2 and has been debugged. It is currently executable to perform the computations. Flow field calculations have not been performed to date since the Telenet link was established on 16 Feb. 1987. So far, during 1986, about an hour of Cray-2 time has been utilized in installing and debugging the code.

Significance:
The development of MDSSL30 code would be of great value in the design and development of transport and fighter aircraft because a greater number of candidate vehicle configurations could be considered and their performance over a wide range of flight conditions could be evaluated, thereby reducing the cost and time required in design cycle.

Future Plans:
Three-dimensional grids will be generated for an MD-80 transport and an F-15 fighter aircraft. Calculations will be performed using fine grids requiring approximately 32 million 64-bit words of Cray-2 memory. The results will be compared with experimental data at transonic Mach numbers.

FULL SIMULATION OF TURBULENCE PAST LARGE EDDY BREAK-UP DEVICES

R. Balasubramanian, Principal Investigator
Cambridge Hydrodynamics, Inc.
NASA Langley Research Center

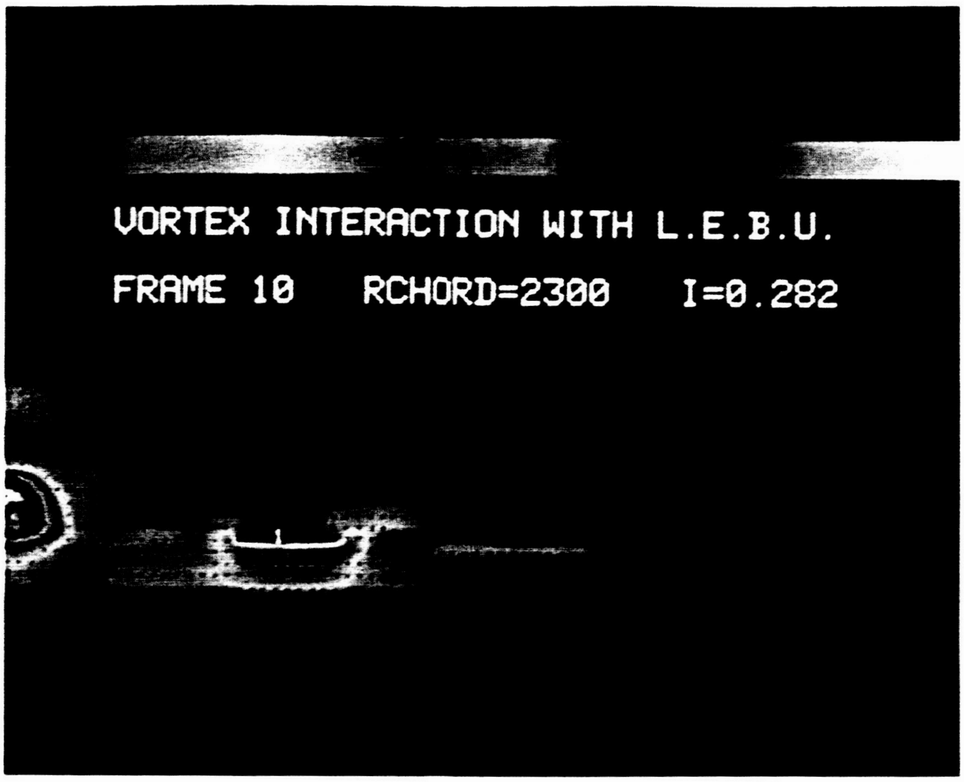
Research Objective:
A full simulation Navier-Stokes Solver for an imbedded body (L.E.B.U.) using inflow-outflow boundary conditions in the flow direction is under development on the Cray-2 Supercomputer. As a precurser to the full simulation, certain parametric studies of the flow problem were undertaken during the IIOC phase. This report attempts to summarize the results of these investigations.

Approach:
The numerical approach uses high order Fourier Chebyshev spectral elements for discretization of the flow domain. In order to obtain C^1 continuity at interelement boundary a Green's function approach is used for spectral patching of domains. The patched spectral elements are ideal for parallel computation since only a limited region of the code is critical (requiring synchronization between CPUs).

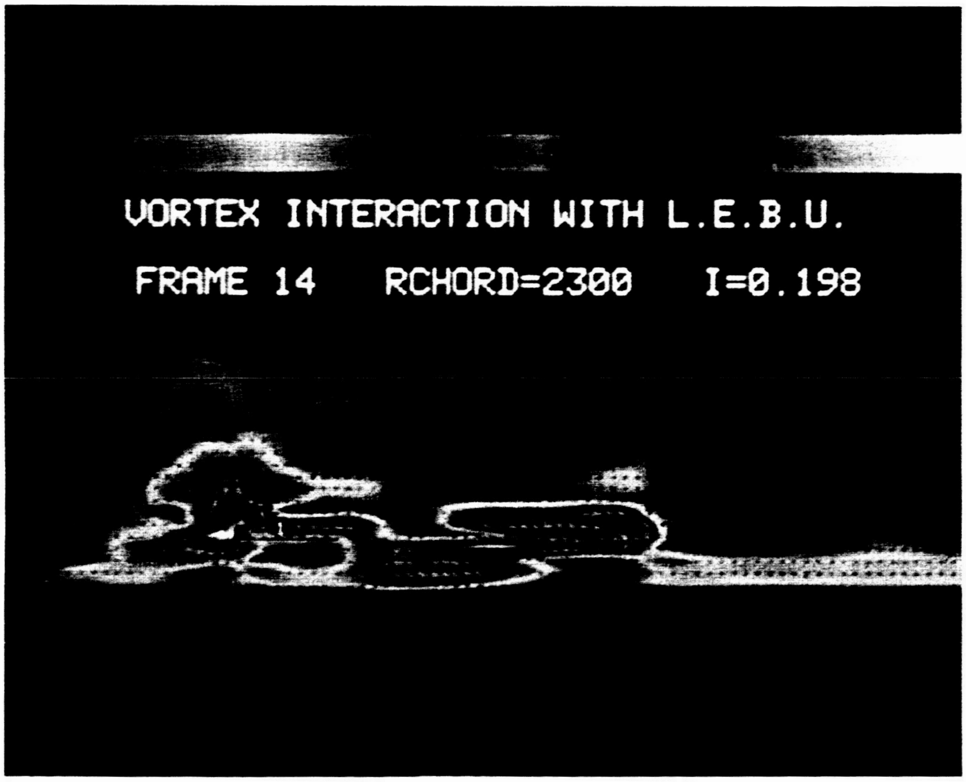
Accomplishment Description:
Experimental investigations on the performance of L.E.B.U. in turbulent boundary layers indicate that the L.E.B.U.s seem to break the large scale structures (turbulent eddies) into smaller scales with an attendant reduction of bursting, and consequently, a lower turbulent drag. As a model problem a numerical experiment was undertaken, consisting of an initially laminar flow past a L.E.B.U. with a known compact vortical structure convecting into the flow domain. The temporal characteristics of this flow were analyzed in detail to understand the effect of L.E.B.U. on this vortical structure. Transverse vortical structures convected into the flow at about the height of L.E.B.U. are segmented into smaller scale structures. The interaction that takes place between the L.E.B.U. and the vortical structure is dependent on the typical chord Reynolds number of the flow. Figures 1,2 show different stages of vortex interaction with L.E.B.U. Longitudinal vortical structures convected into the flow field were also studied with and without the L.E.B.U. structures. For the case of a flow without the L.E.B.U. the longitudinal structures drift toward the wall and create counter-vorticity at the wall. The counter-rotating structures drift upward at later times because of net upward velocity. This effect has been experimentally observed (Didder & Ho, J.F.M., 1985) for a jet flow impinging into a wall. The presence of L.E.B.U. was shown to produce substantial flow interactions. Raking the flow field at various locations, the streak line pattern was observed to show that the presence of L.E.B.U. reduces the vorticity for all cases that were sampled, compared to the flow without the L.E.B.U.s. The effect of $v.n=0$ on the L.E.B.U. traps a portion of the vortical structure under the L.E.B.U., creating a small scale pair of counter-rotating vortices that eventually merge with the top half vortex at streamwise planes downstream of the L.E.B.U., and creates complicated vortical amalgamation patterns that eventually exit the computational domain.

Significance:
Patched elements allow analysis of complicated flow geometries. A new approach using patching would enable fully developed turbulent flow simulation at moderately high Reynolds numbers.

Future Plans:
Further work in the area is anticipated.



BALASUBRAMANIAN: FULL SIMULATION OF TURBULENCE PAST LARGE EDDY BREAD-UP DEVICES



BALASUBRAMANIAN: FULL SIMULATION OF TURBULENCE PAST LARGE EDDY BREAD-UP DEVICES

FULL NAVIER-STOKES SOLUTION METHODOLOGY

Stan Birch, Principal Investigator
Co-investigators: Robert Blakeley, Joe Hoffman, Eric Holcomb, Robert Hopcroft, and Patrick Rodi
Boeing Aerospace Company

Research Objective:
Development of a technology base (including geometry descriptors, mesh generators, turbulence modeling, numerical algorithms and pre- and post-processing) which supports a state-of-the-art three-dimensional full Navier-Stokes flow prediction capability.

Approach:
The code is being developed in a modular approach. Each of the technology elements listed above will be easily modified and extended to take advantage of new developments in any of the areas or in computer hardware.

Accomplishment Description:
The progress made during the IIOC phase represents the first operation of our N/S code on a Cray-2 supercomputer as well as the first operation of our code on any NASA-Ames computer system. The major accomplishments made during the IIOC phase include transfer, modification and testing of our N/S code and our grid generator. In general, our experience shows that the Cray-2 is about 20% slower than Boeing's Cray-XMP-24. Incorporating Cray-2 specific enhancements (e.g., high speed local memory) should significantly diminish the CPU time needed. Other enhancements (e.g., microtasking, macrotasking) should decrease the elapsed wall clock time.

Significance:
The above accomplishments represent a major improvement of the computer capabilities available to Boeing. The successful transfer and operation of our N/S code demonstrates the code's built-in portability. This Navier-Stokes code, combined with the NAS system, represents a powerful analysis capability available for a variety of practical problems.

Future Plans:
The enclosed photo shows the nose section of a 747SP aircraft with a large "cavity" opening on the copilot's side of the fuselage. The Cray-2 will be used to calculate this highly three-dimensional cavity problem at the cruise flight conditions of $M_\infty = 0.84$ and $h = 41,000$ ft. The resulting Navier-Stokes solutions will be analyzed by ray tracing programs for the detailed understanding of the aero-optical properties of this configuration.

TURBULENT FLOW

Arthur D. Carlson, Principal Investigator
Craig A. Wagner, Associate Investigator
Engineering Mechanics Division
Naval Underwater Systems Center

Research Objective:
To develop numerical capability for computing laminar and fully turbulent incompressible flow.

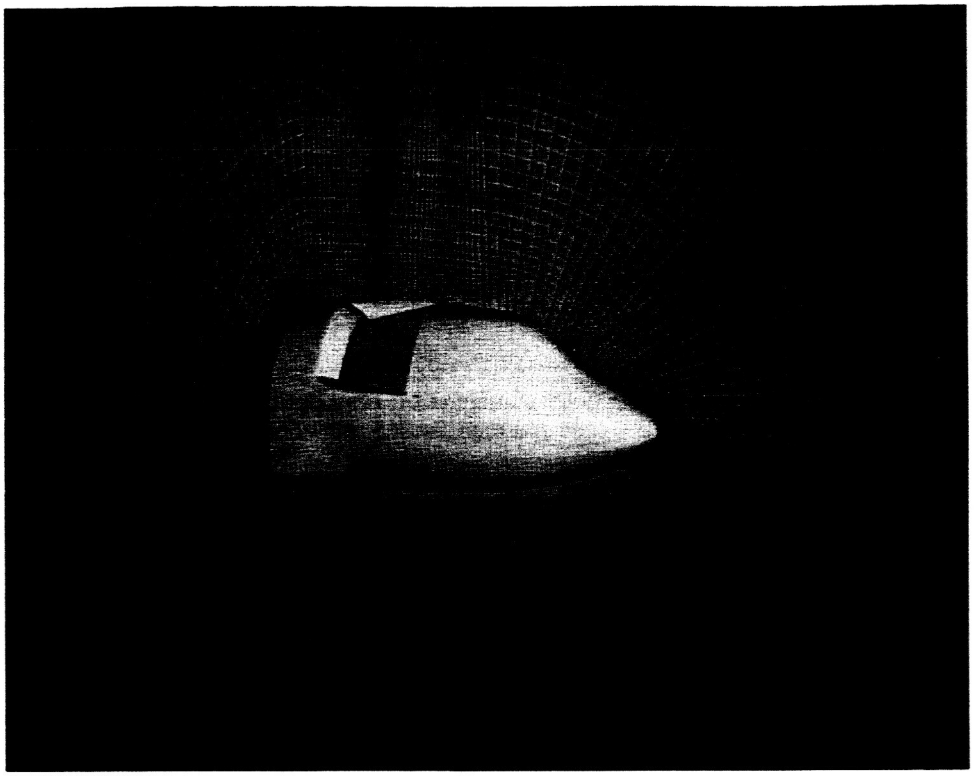
Approach:
The INS3D computer code developed at NASA Ames Research Center was extended to include steady, turbulent, incompressible flow fields. Turbulent models incorporated in INS3D include a simple algebraic eddy viscosity model and a two-equation k–ϵ model. Results using these models were checked against a valid experimental data base.

Accomplishment Description:
The two turbulence models implemented in INS3D were checked against experimental measurements of flow around an endwall mounted cylinder at Reynolds number 550,000 taken by Eckerle and Langston. To ensure accuracy of the numerically predicted flow fields, a refined mesh was used which included a flow domain of 559,944 finite difference points. Total Cray-2 storage was slightly greater than 20 million words. The algebraic eddy viscosity model required eight Cray-2 hours per run and the k–ϵ turbulence model eleven hours per run. Initial quantitative results indicate a number of important features: (1) there appears to be very little difference in the flow field computed by the algebraic model versus the k–ϵ model, (2) vortex dynamics associated with the roll-up and horseshoe vortex in front of the cylinder appears to be model independent for the two turbulence models used here. Another result relates to the behavior of the shear stress gradients at the saddlepoint of separation in front of the cylinder. Wagner used the Oswatitsch model to describe this separation process. However, as observed by Langston at the University of Connecticut, the results from ten laminar computer runs and the two turbulent runs indicate the separation process actually involves shear stress gradients at the wall opposite in sign to that which is currently believed to be correct. This may have impact for understanding separation processes. Figure 1 shows particle traces in the horseshoe vortex system for the endwall mounted cylinder at Reynolds number 550,000.

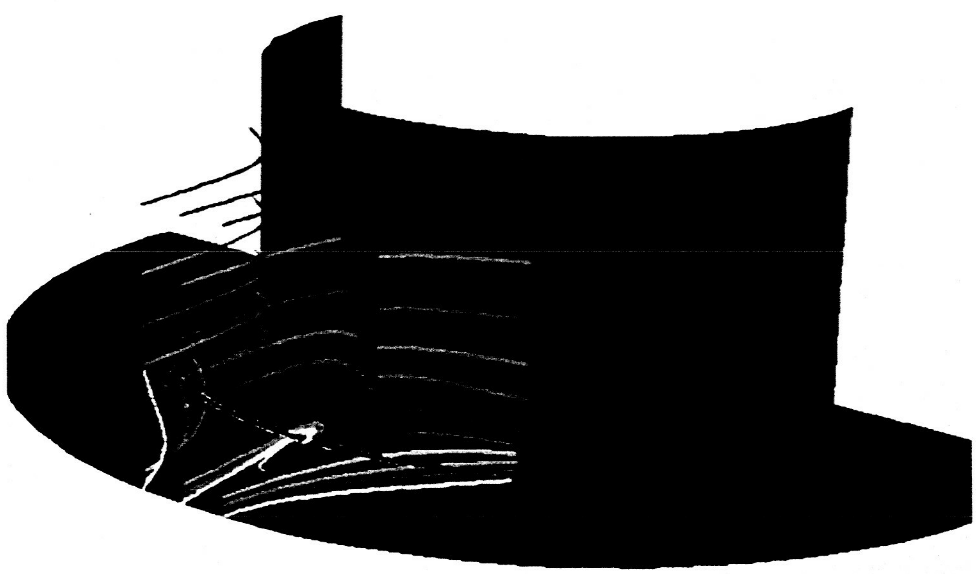
Significance:
The validation of the revised code and verification of two turbulence models for this highly separated flow demonstrate the feasibility of analyzing steady, turbulent flow fields.

Future Plans:
Continue the study of turbulence models to choose the most relevant to our particular problems and add time-dependent mesh generation and transient flow analysis capabilities.

Publications:
Wagner, C. A., Ph.D. Thesis, University of Connecticut, Storrs, Connecticut, to be completed June 1987.



BIRCH: FULL NAVIER-STOKES SOLUTION METHODOLOGY



CARLSON: TURBULENT FLOW

HYPERSONIC FLOW PAST GENERIC LIFTING BODIES

Denny S. Chaussee, Principal Investigator
Co-investigators: Yehia Rizk and Joe Steger
Sterling Federal Systems Inc./NASA Ames Research Center

Research Objectives:
Compute the flow field around a generic wing body configuration at hypersonic speeds.

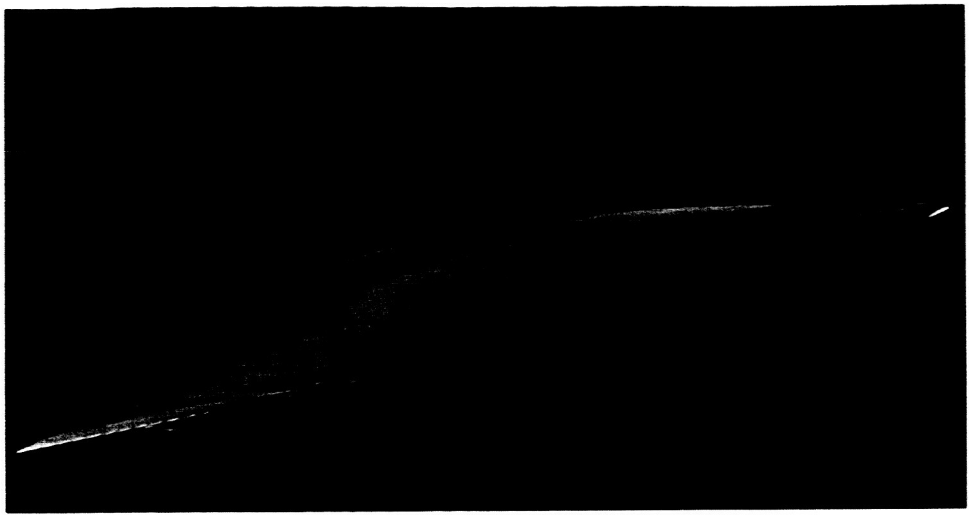
Approach:
Use the 3D hyperbolic grid generation scheme to create a single grid around the whole configuration and then solve the flow field in one or more segments using the partially upwind scheme (F3D).

Accomplishment Description:
Computations were performed for a generic wing body configuration at Mach number 25 and 5° angle of attack. This computation required about 10 hours of the Cray-2 CPU time and used about 5 million words of the Cray-2 core memory.

Significance:
The computations provide the designer with important quantities such as the surface stress and heat transfer. This shows how useful CFD is in the design process, especially when wind tunnels cannot simulate the actual hypersonic flight conditions.

Future Plans:
Perform the computations for a more realistic shape and add real gas effects to the code.

Publications:
Rizk, Y., Chaussee, D., and Steger, J.: Numerical Simulation of the Hypersonic Flow around Lifting Vehicles. AGARD Symposium on Aerodynamics of Hypersonic Lifting Vehicles, Bristol, England, Apr. 6-9, 1987.



CHAUSSEE: HYPERSONIC FLOW PAST GENERIC LIFTING BODIES

SUPERCRUISE FIGHTER FLOW SIMULATION

Denny S. Chaussee, Principal Investigator
Co-investigator: R. Uwe Jettmar
NASA Ames Research Center

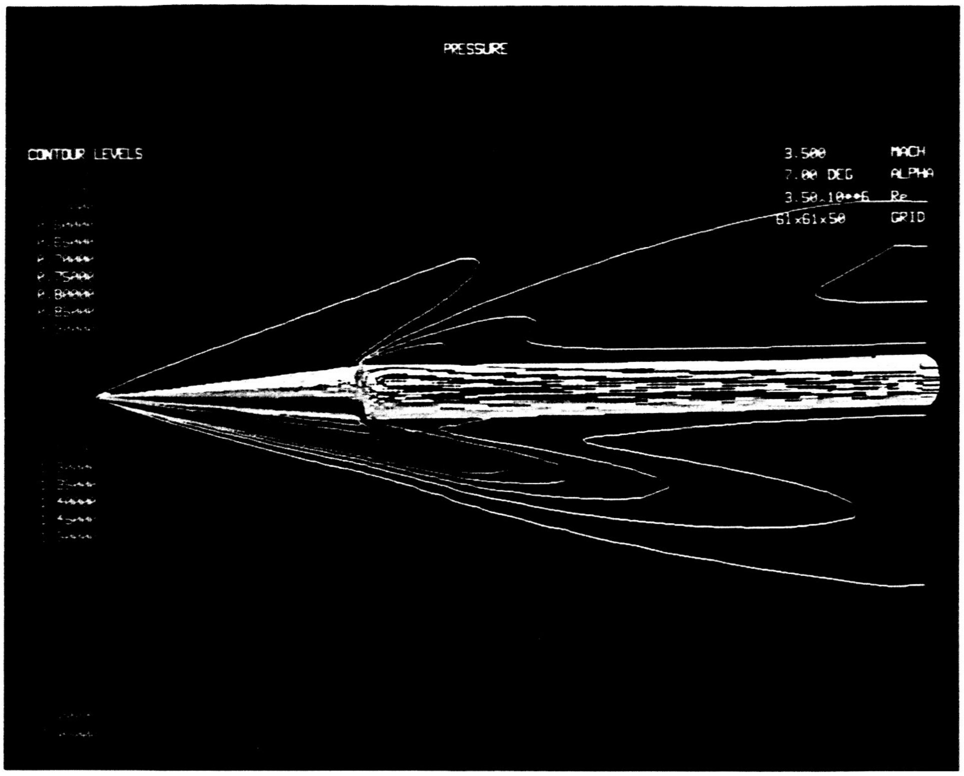
Research Objective:
A combined computational and experimental study of supersonic flow over body-wing configurations.

Approach:
Flow over a cone-cylinder and ogive-cylinder have been generated for $M = 3.5$ and various angles of attack. An algorithm featuring two implicit factors and partial flux splitting is used to solve the thin-layer Navier-Stokes equations.

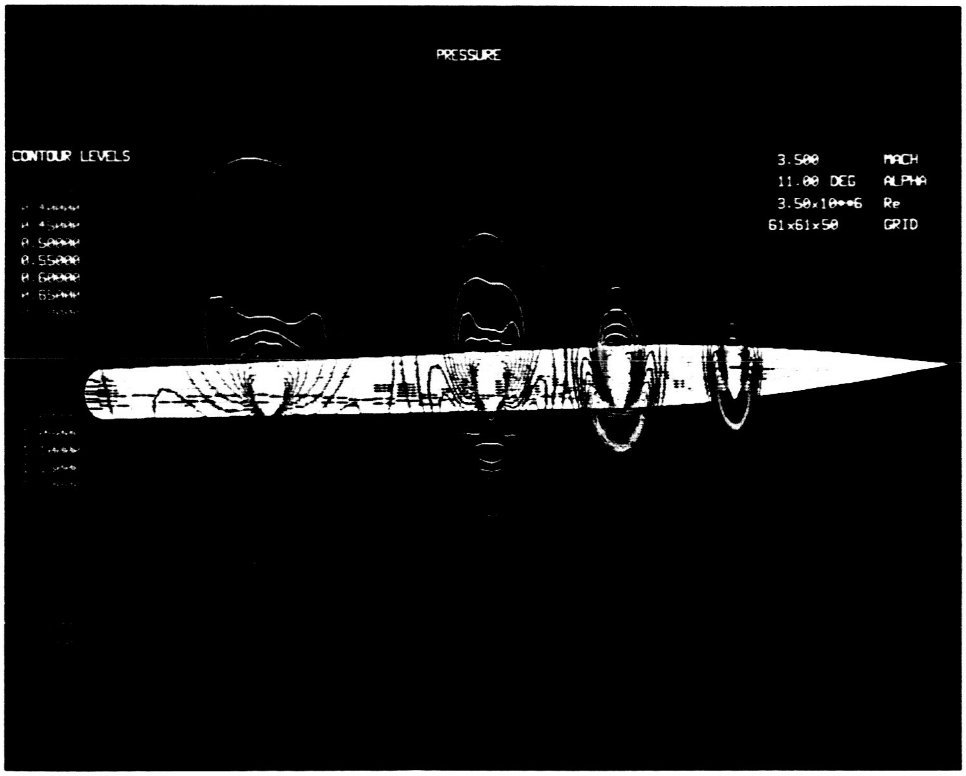
Accomplishment Descriptions:
In the present study pressure distributions for a slender long cone-cylinder and ogive-cylinder have been generated with the goal to become familiar with the computational facilities of the Cray-2 as well as the solution technique, the F3D code. These particular configurations were chosen for the existence of experimental results. The data transmitted to RAF did not allow for easy comparison, because length scales on the data were missing to establish a qualitative comparison between experiment and computation. However, the computations resulted in a qualitative agreement with the experiments. They are shown in the attached figures for cone-cylinder at $M = 3.5$ and $7°$ angle of attack and for an ogive-cylinder at $M = 3.5$ and $11°$ angle of attack.

Significance:
The results obtained show the feasibility of computational flow simulation for practical configuration using 3-D Navier-Stokes codes on the Cray-2.

Future Plans:
No future plans are anticipated.



CHAUSSEE: SUPERCRUISE FIGHTER FLOW SIMULATION



CHAUSSEE: SUPERCRUISE FIGHTER FLOW SIMULATION

TRANAIR EVALUATION

Allen W. Chen, Principal Investigator
Boeing Commercial Airplane Company

Research Objective:
To obtain the effects of grid density on the TRANAIR solutions.

Approach:
Run TRANAIR for a sample case, a 747 wing/body configuration, with different grid densities.

Accomplishment Description:
Made one successful run which used the coarse grid 126X41X26. The surface pressure distribution is presented in the attached photo. Ten million word memory and 3,000 seconds of CPU time were used.

Significance:
The run illustrated the necessity for local grid embedding which is now being implemented.

Future Plans:
We will get back to the evaluation when the code developers have sped up the program.

AIRFRAME/INLET AERODYNAMICS

Wei J. Chyu, Principal Investigator
Co-investigators: T. Kawamura and Greg Howe
NASA Ames Research Center

Research Objective:
Develop efficient numerical techniques for airframe/inlet aerodynamic analysis.

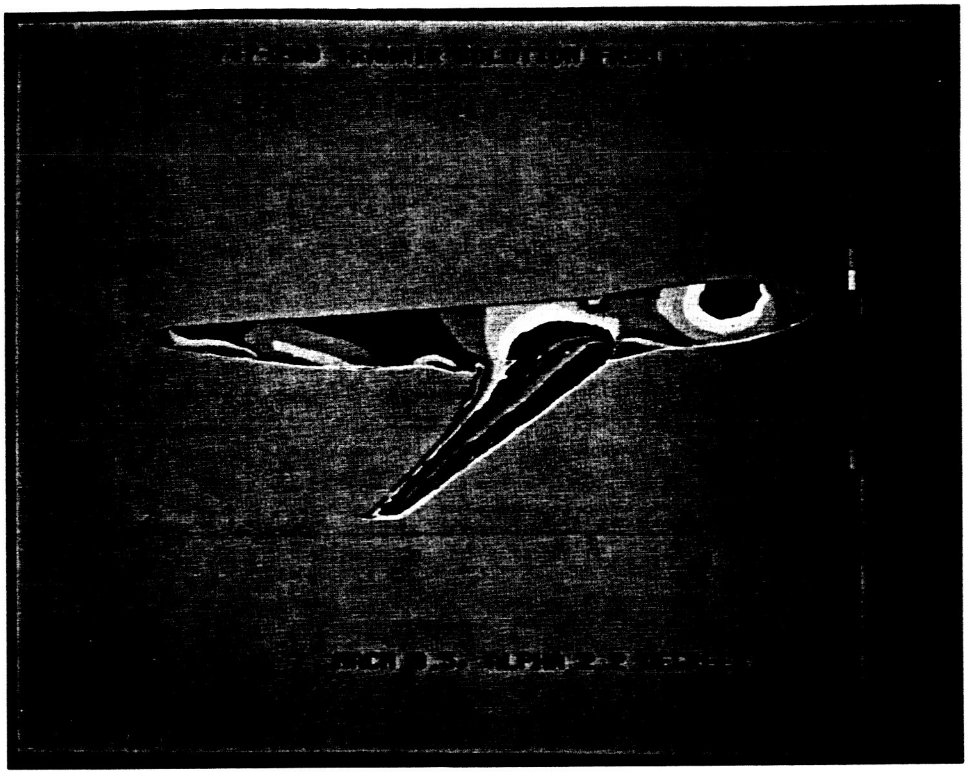
Approach:
Numerically compute the flow fields by solving the Navier-Stokes equations.

Accomplishment Description:
A computer code incorporated with a combined implicit-explicit method for solving the Navier-Stokes equations is developed, and used for computing supersonic inlet flows with mixed external-internal compressions of an axisymmetric inlet model. The computed flow fields depict shock-wave intersections, flow spillage around the cowl lip, shock-wave/boundary layer interactions, internal normal (terminal) shocks, and the effects of flow incidence. Computed results are compared well with available wind tunnel data.

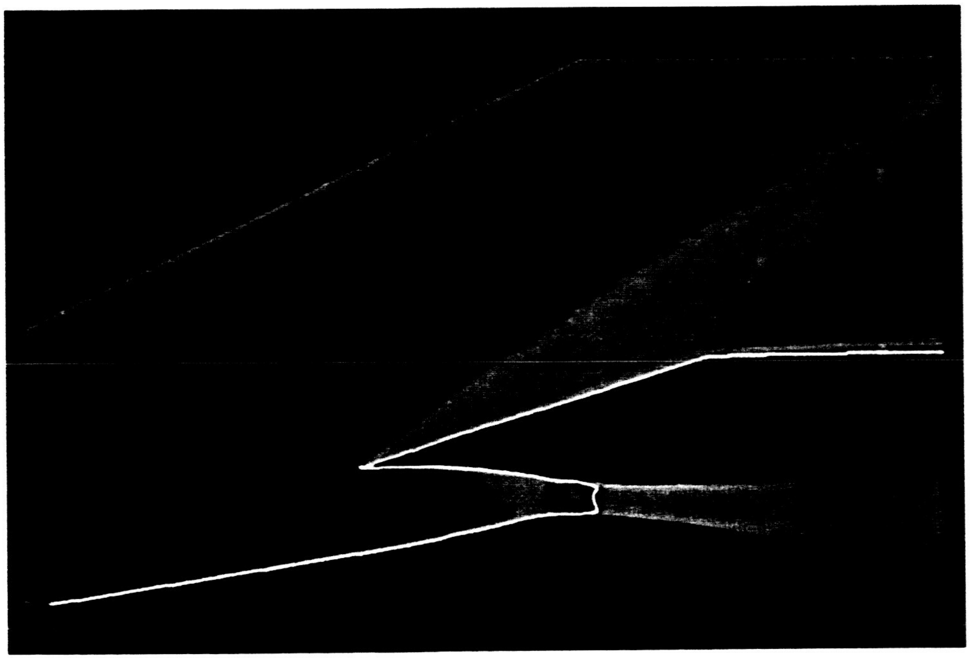
Significance:
Computational techniques were successfully developed for the computation of 3-D mixed compression inlet flow fields. The results have contributed to further understanding of the inlet flow physics and provided new methodology for the computations of overall (internal/external) flow fields.

Future Plan:
1) Extend the work by adding a nozzle on the inlet model to simulate the engine-face boundary condition.
2) Improve the accuracy by applying a partially flux-split Navier-Stokes code to the inlet flow computations.

Publications:
AIAA Paper 87-0160, NASA TM 88362.



CHEN: TRANAIR EVALUATION



CHYU: AIRFRAME/INLET AERODYNAMICS

DIRECT NUMERICAL SIMULATION OF CHEMICALLY REACTING FLOWS

R. W. Claus, Principal Investigator
NASA Lewis Research Center

Research Objective:
Study the basic hydrodynamic mechanisms that lead to augmented mixing in chemically reacting flows.

Approach:
Using highly accurate numerical methods (i.e., spectral methods), simulate the temporal evolution of a mixing layer subject to various types of forcing.

Accomplishment Description:
Numerical simulations of a mixing layer subject to various types of forcing have been conducted with greater resolution than has previously been possible. This greater resolution allows calculations at higher Reynolds numbers – resulting in significant changes in the vortical evolution of the flow. These changes in the vortex structure suggest turbulence control strategies to enhance turbulent mixing. If these schemes prove effective, very low amplitude forcing may be able to significantly alter chemically reacting flows.

Each calculation on the Cray-2 required about 15 million words of core storage and approximately 40 CPU hours. About 200 total CPU hours were used.

Significance:
Turbulence control strategies may be useful in achieving stable, efficient combustion in the NASP propulsion system.

Future Plans:
Additional studies of turbulent mixing at high Reynolds number are planned.

Publications:
Direct Numerical Simulations of a Temporally Evolving Mixing Layer Subject to Forcing.

INVESTIGATION OF THE TURBULENT AMPLIFICATION OF MAGNETIC ENERGY IN AN INCOMPRESSIBLE MAGNETOFLUID

Russell B. Dahlburg, Principal Investigator
Co-investigator: Jill P. Dahlburg
Naval Research Laboratory

Research Objective:
To investigate the viscoresistive decay of turbulent, three-dimensional magnetofluids with initial, finite magnetic helicity.

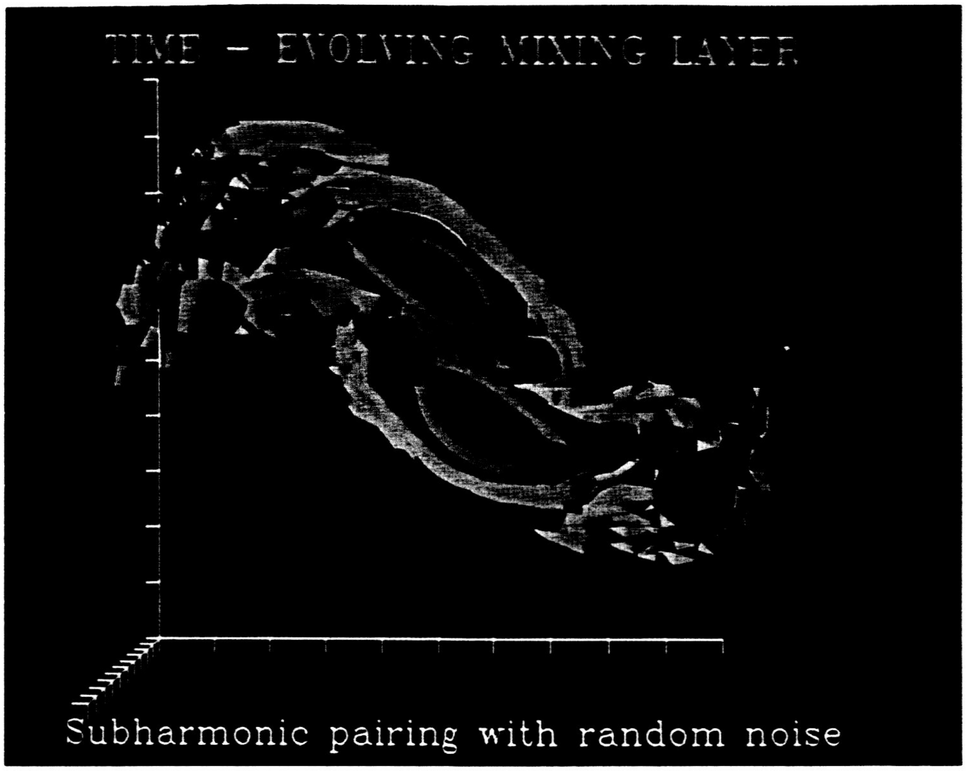
Approach:
Direct numerical simulation of the three-dimensional, viscoresistive, incompressible, magnetohydrodynamic equations in a periodic geometry. Fourier pseudospectral discretization is employed in all three spatial directions. Time is discretized by a second-order Runge-Kutta scheme.

Accomplishment Description:
We have successfully modeled the decay of the helically turbulent magnetofluid. The decay of the magnetofluid is characterized by the condensation of the magnetic energy in the largest spatial scales of the system, and the condensation of the electric current density and vorticity in the shortest spatial scales of the system. This structural behavior, combined with the selective decay of the total energy with respect to the magnetic helicity as a function of time, implies that the magnetofluid is self-organizing. A typical run requires 5–10 hours and 1.3 million words of storage.

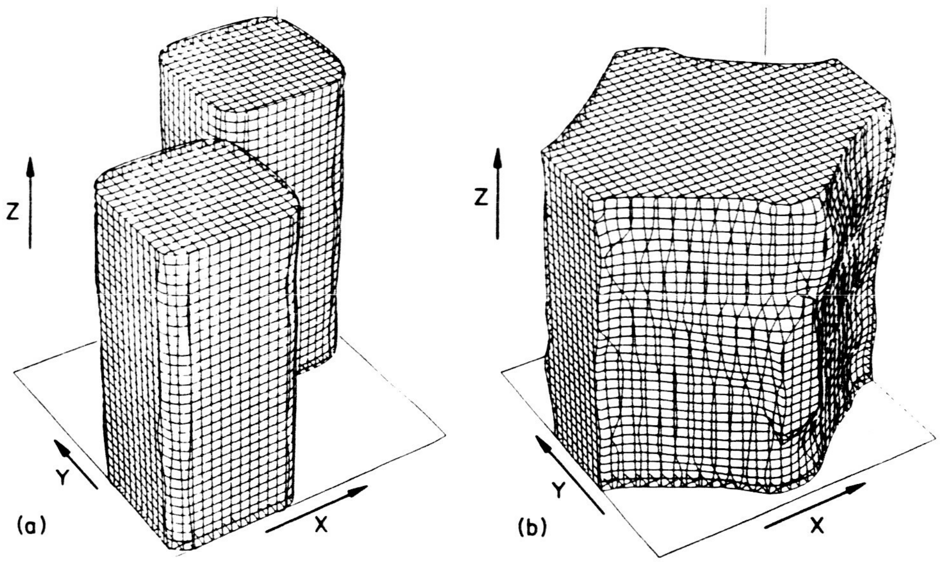
Significance:
The magnetofluid behavior described above is believed to occur in the heating of the solar corona by local current production. Observations of the magnetic field and velocity field in the solar corona are very imprecise. Hence, numerical simulation of this decay process is required to increase our understanding.

Future Plans:
1) Alter one spatial direction to allow for nonperiodic boundary conditions in the numerical algorithm.
2) Add compressibility effects to the governing equations.

Publications:
R. B. Dahlburg and J. P. Dahlburg: Numerical Simulation of 3-D MHD Turbulence. Bull. Am. Phys. Soc., vol. 31, 1986, p. 1514.
R. B. Dahlburg, J. P. Dahlburg, and J. T. Mariska: Numerical Simulation of Turbulent Coronal Self Organization. Bull. Am. Astron. Soc., vol. 18, 1986, p. 901.



CLAUS: DIRECT NUMERICAL SIMULATION OF CHEMICALLY REACTING FLOWS



Evidence of turbulent self–organization in a three–dimentional magnetofluid with a high level of magnetic helicity present initially. The figure shows contours of negative z magnetic field at *a.* t = 0.5 and *b.* t = 53.0, where time is measured in units of Alfven transit times. The condensation of the magnetic energy at the largest spatial scales in the system is expected to occur as the magnetofluid self-organizes.

DAHLBURG: INVESTIGATION OF THE TURBULENT AMPLIFICATION OF MAGNETIC ENERGY IN AN INCOMPRESSIBLE MAGNETOFLUID

HELICOPTER ROTOR BLADE DESIGN USING FULL POTENTIAL/WAKE METHODOLOGY

T. Alan Egolf, Principal Investigator
United Technologies Research Center

Research Objective:
Demonstrate both the merits of using advanced aerodynamic codes for the prediction of airloads and acoustic properties of rotor blades and the remote site use of the NAS to the helicopter industry as endorsed by the American Helicopter Society.

Approach:
Predict the region of supersonic flow extending beyond the blade tip relative to the blade fixed coordinate system (delocalization region) for various rotor blade geometries including the effects of "realistic wake geometries" using the ROT22/WAKE full potential analysis.

Accomplishment Description:
It was shown that the inclusion of "realistic wake effects" had a significant impact on the size of the delocalization regions of importance for helicopter rotor noise. The analysis was applied to rotor blade designs which had been previously "optimized" (neglecting wake effects) in planform and section profile design to reduce the extent of the delocalization region. Rectangular and swept blade designs were studied using both two- and four-bladed rotor configurations including classical undistorted and generalized wake (a form of distorted wake) geometry models, along with the original model (no wake influence). Approximately 10 final cases were run with an average Cray-2 run time of 1.5 hours per case and a memory requirement of 1.2 MW. A total of about 20 hours were used during this study. As part of this activity, the results were downloaded from the Cray-2 to a local UTRC color workstation using the NAS Long Haul Communications System (LHCS) to view the delocalization regions as part of the demonstration of the use of NAS to the helicopter industry.

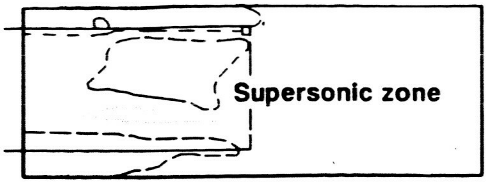
Significance:
This study demonstrated the sensitivity of the prediction of the acoustically important delocalization region to the influence of the rotor wake, and the need to include accurate wake geometry models in future rotor blade design optimization studies. This activity also represented the first remote site application of NAS for helicopter CFD codes.

Future Plans:
Continue the use of NAS to develop, demonstrate, and evaluate computationally intensive rotary-wing CFD codes for airload and performance prediction, and perform remote-site demonstrations for the helicopter industry.

Publications:
Anton J. Landgrebe: Remote Site Demonstration of Helicopter CFD Codes. Status Report to National Specialist's Meeting on Aerodynamics and Acoustics, American Helicopter Society, February 1987.
Anton J. Landgrebe: Computational Fluid Dynamics Research at the United Technologies Research Center Requiring Supercomputers. NASA Conference on Supercomputing in Aerospace, March 10-12, 1987.

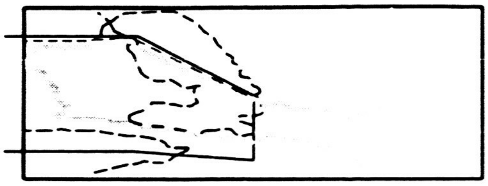
NAS REMOTE SITE DEMONSTRATION

Sonic delocalization from helicopter blade tip



Rectangular blade-generalized wake, B=4

- **NASA ARC/UTRC ROT 22/WAKE code (full potential theory)**
- **NAS supercomputer - remote from UTRC**



Swept blade-generalized wake B=4

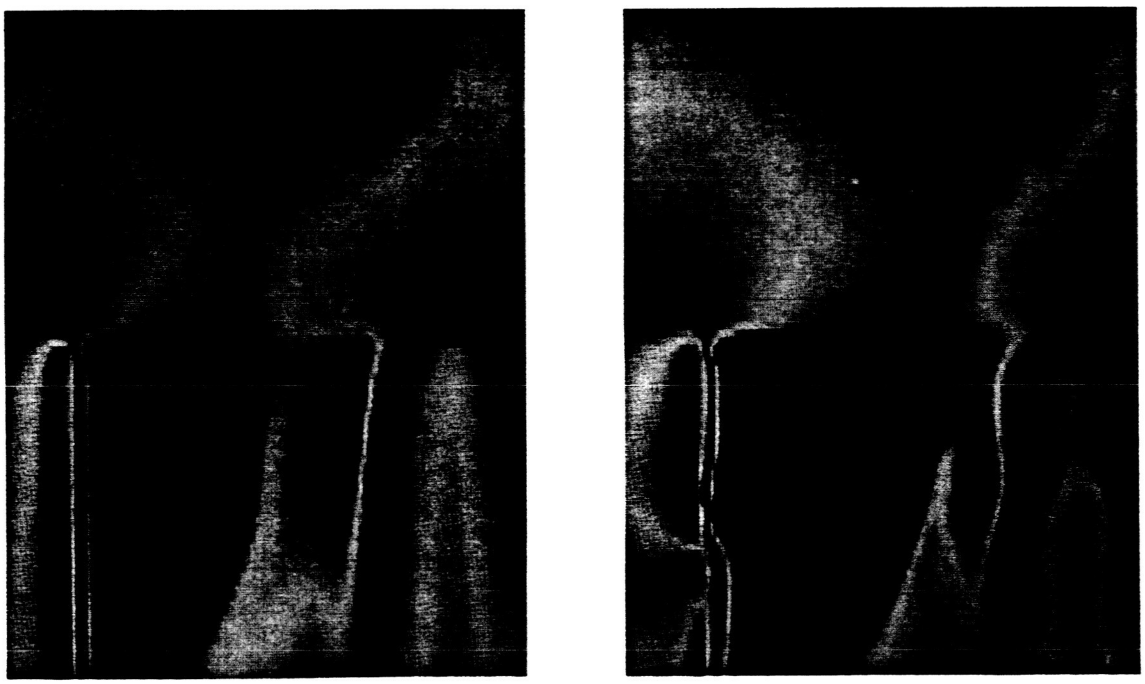
Swept blade-flat wake, 1/4 rev

UNITED TECHNOLOGIES RESEARCH CENTER

RA1956TX.003

EGOLF: HELICOPTER ROTOR BLADE DESIGN USING FULL POTENTIAL/ WAKE METHODOLOGY

SONIC DEVELOPMENT FROM HELICOPTER BALDE TIP



Mach number distributions on the plane defined by the upper surface of the rotor blade and its extension onto the plane of rotation are shown from the outer 90% of the blade to 110% of the blade for a rectangular blade. The left-hand picture shows the results for the case where no wake influence is used in the analysis and the right-hand picture shows the results for the four-bladed generalized wake model. The presence of a strong tip vortex is seen in the right-hand picture. The extent of the delocalization region off the tip of the blades is shown to be larger in the prediction obtained using the wake model. The leading edge of the blade is near the dark blue region on the left-hand side of each picture and the trailing edge is in the light blue area on the right-hand side of each picture. The tip of the blade is midway down each picture. Dark blue represents the lowest Mach Number, while red is the highest. Pure yellow is the sonic value, and it shouldbe noted that the color scale is not linear with Mach Number.

EGOLF: HELICOPTER ROTOR BLADE DESIGN USING FULL POTENTIAL/ WAKE METHODOLOGY

COMPRESSIBLE TRANSITION

Gordon Erlebacher, Principal Investigator
Co-investigator: M. Y. Hussaini
NASA Langley Research Center

Research Objective:
To study the mechanisms underlying the transition of wall-bounded flows at low and high Mach numbers and predict its onset.

Approach:
Direct simulations of the initial stages of transition in compressible flows are performed by numerically solving the full, three-dimensional, time-dependent compressible Navier-Stokes equations. Spectral collocation methods are used in all directions to insure a high degree of accuracy.

Accomplishment Description:
Starting from a mean boundary-layer profile (solution to the compressible similarity equations) and two Tollmien-Schlichting (TS) waves (eigenfunctions of the linearized Navier-Stokes equations), the initial stages of K-type breakdown were simulated. The Mach number is 4.5 and the Reynolds number based on displacement thickness is 10000 (to insure an unstable initial wave). The waves chosen correspond to the first mode among the multiple modes present in compressible flows. Compressibility is found to have a modifying effect on the onset of the secondary instability when compared to similar incompressible simulations. For example, starting with perturbation amplitudes of 0.4% and 1.5% for the 3-D and 2-D waves, respectively, the instability barely made an appearance after 9 TS time periods. Amplitudes three times as high resulted in the onset of the instability after approximately 5 periods. Experiments were also performed in which the lowest order spanwise Fourier mode was artificially suppressed. As for incompressible flows, the initial stages of the K breakdown was retarded. The code is currently explicit. On a 48X36X65 grid, memory requirements did not exceed 3 Mwords, but the time necessary to reach 9 TS periods exceeded 100 CPU hours. Over 400 hours were used in the pilot period, and 100 hours in the IIOC. The code is fully vectorized and runs at a speed greater than 100 Mflops.

Significance:
An understanding and prediction of transition in high-speed flows will greatly influence the design of key components in future high-speed transports.

Future Plans:
Continue the study of the secondary instability at $M = 4.5$ with an implicit version of the code and develop transition criteria.

Publications:
Erlebacher, G.: Transition Phenomena Over a Flat Plat for Compressible Flows. Tenth International Conference on Numerical Methods in Fluid Dynamics. Lecture Notes in Physics, vol. 264, 1986, pp. 265–269.
Erlebacher, G., and Hussaini, M. Y.: Incipient Transition Phenomena in Compressible Flows Over a Flat Plate. ICASE Report No. 86-30, NASA CR-178111, 1986.
Zang, T. A., Drummond, J. P., and Erlebacher, G.: Numerical Simulation of Transitions, Compressible Turbulence, and Reacting Flows. AIAA Paper No. 87-0130, 1987.

NAVIER-STOKES SIMULATION OF HYPERSONIC REAL GAS FLOW FIELDS

William J. Feiereisen, Principal Investigator
Co-investigator: Etheraj Venkatapathy
NASA Ames Research Center

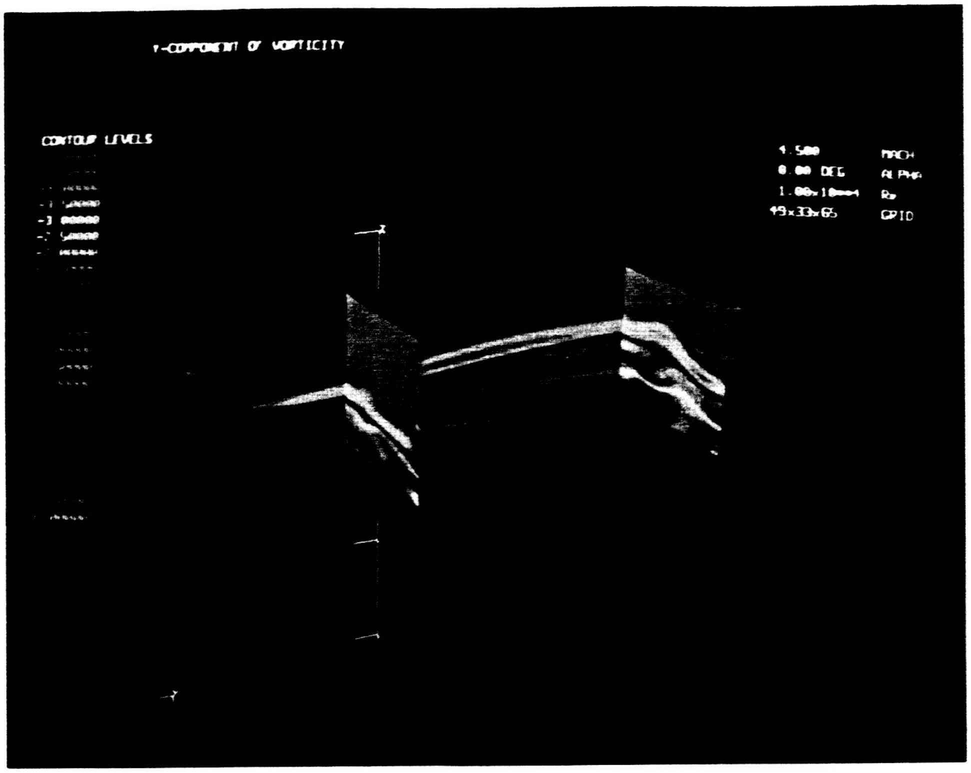
Research Objective:
To develop the capability to simulate 3-D flow fields and the high temperature chemistry around bodies at hypersonic Mach numbers.

Approach:
A flux split algorithm for the Navier-Stokes equations has been combined with a method for calculating realistic quantities in air in equilibrium at high temperature.

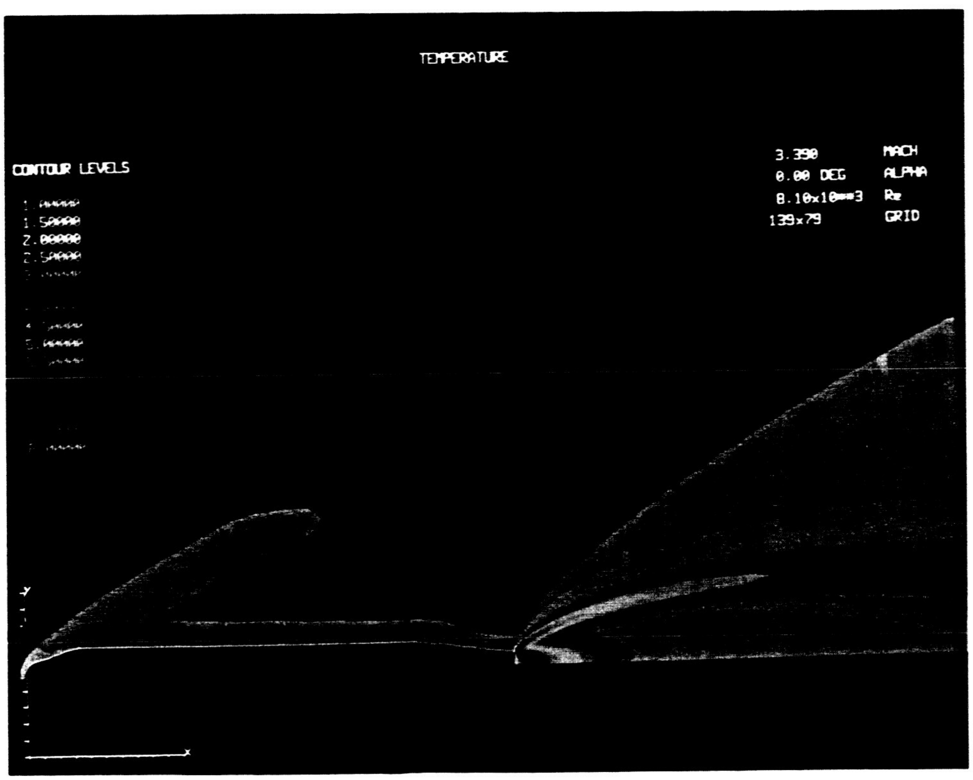
Accomplishment Description:
The method has been developed to calculate viscous flow fields with equilibrium air chemistry. The specific geometry used as a test case so far has been the "body/plume." Calculations have been successfully performed at external Mach numbers up to 20. The accompanying graphic shows a configuration at lower Mach number with a hot jet exiting the nozzle. Approximately twelve CPU hours have been used at this time. The memory used depends upon the mesh size and number of species and ranges from 3 to 20 MW.

Significance:
A robust code to calculate hypersonic real gas flow fields has been developed that will serve as a vehicle for the implementation of nonequilibrium chemistry models.

Future Plans:
Implementation of nonequilibrium chemistry models. Adaptive gridding to improve resolution. Enhancement of the underlying data management system to allow more flexible geometry definition.



ERLEBACHER: COMPRESSIBLE TRANSITION



FEIEREISEN: NAVIER-STOKES SIMULATION OF HYPERSONIC REAL GAS FLOW FIELDS

PARTICLE SIMULATION OF COMPRESSIBLE FLOW

William J. Feiereisen, Principal Investigator
Co-investigators: Jeffrey McDonald and Prof. D. Baganoff
NASA Ames Research Center/Stanford University

Research Objective:
Study of hypersonic flows with chemical nonequilibrium effects.

Approach:
Direct particle simulations employing a new efficient collision algorithm.

Accomplishment Description:
The accompanying photo depicts an ideal, infinite Mach number flow impinging on a flat plate. Each color represents a density level, the scale of which is shown in the bottom left corner of the figure. The theoretical density level for the stagnation region as well as the free stream density are marked on the scale. This simulation utilized about 34,000 particles tracked for over 15,000 time steps. Each particle in the simulation has five degrees of freedom, three translational and two rotational. Approximately three and a half CPU hours and 33 MW were needed for this run.

Significance:
Complex flow solutions agree well with theory. Method provides solutions in heretofore difficult flow regime.

Future Plans:
A model for vibrational mode is being developed. Larger scale problems and more complex geometries will be studied.

THE STUDY OF THREE-DIMENSIONAL VORTEX DYNAMICS USING A VORTICITY-VELOCITY NUMERICAL ALGORITHM

Thomas B. Gatski, Principal Investigator
Co-investigators: Chester E. Grosch and Raad A. Fatoohi
NASA Langley Research Center

Research Objective:
Optimization of unique three-dimensional vorticity-velocity algorithm for efficient solution of vortex dominated flows.

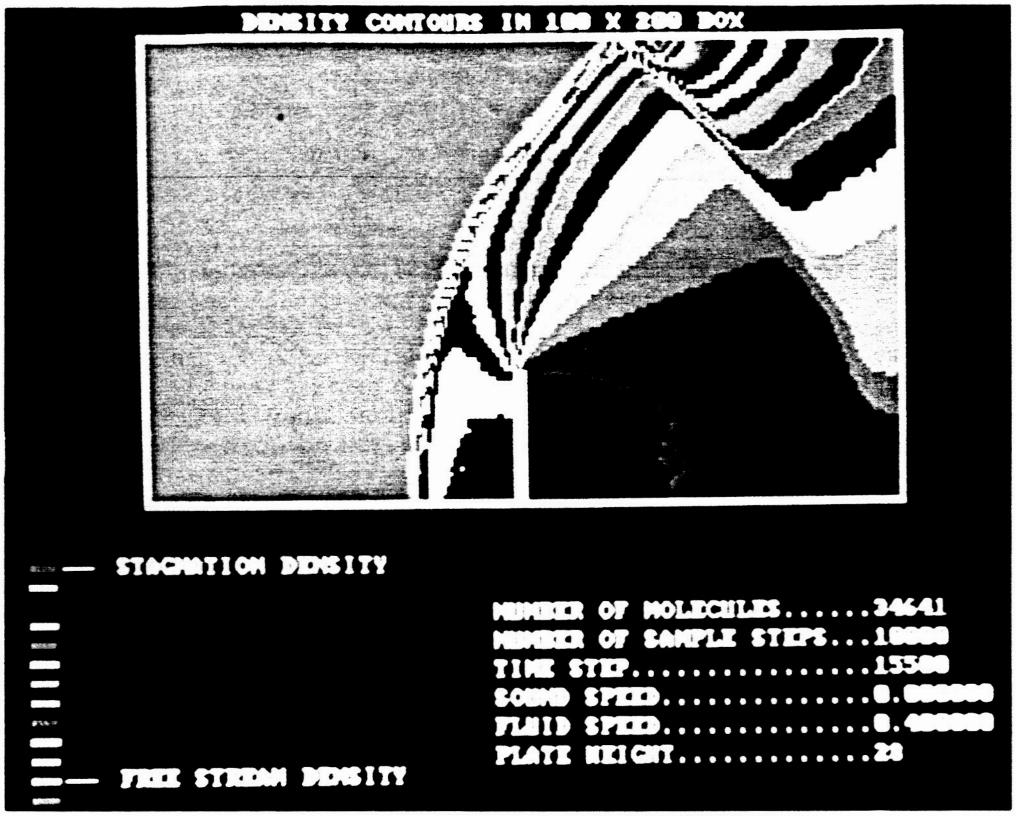
Approach:
Program code restructuring to best utilize Cray-2 architecture.

Accomplishment Description:
The major effort has been directed toward adapting the three-dimensional, time dependent, complete Navier-Stokes code for an incompressible fluid to the Cray-2. The code was originally developed to be run on a serial computer, without serious consideration of vectorization. When run through the CFT compiler only a few of the inner loops of the code were vectorized. An average processing rate of about 30 megaflops was obtained. The typical segment of the code is a relaxation sweep over the three-dimensional domain. A typical segment was extracted from the code and used for development. After restructuring the code, using compiler directives, and using library functions the processing rate was increased from 30 megaflops to over 100 megaflops. The complete code is now being used to study the time evolution of three-dimensional vortex.

Significance:
Efficient and accurate solution of physically relevant vortex dominated flows using the vorticity-velocity algorithm.

Future Plans:
Apply algorithm to such problems as vortex breakdown and leading-edge receptivity effects.

Publications:
R. A. Fatoohi and C. E. Grosch: Proceedings of the 1987 International Conference on Parallel Processing, sponsored by IEEE, ACM, and Penn State University, August 1987.



FEIEREISEN: PARTICLE SIMULATION OF COMPRESSIBLE FLOW

LANGLEY AEROTHERMODYNAMIC UPWIND RELAXATION ALGORITHM (PROGRAM LAURA)

Peter A. Gnoffo, Principal Investigator
NASA Langley Research Center

Research Objective:
Develop and improve numerical methods capable of modeling hypersonic, nonequilibrium gas flows with emphasis on supporting Aeroassisted Orbital Transfer Vehicles.

Approach:
Finite-volume, point-implicit, Symmetric Total Variation Diminishing (STUD) discretization of Navier-Stokes equations.

Accomplishment Description:
Program LAURA has been converted to Cray-2 compatible, vectorizable Fortran 77 code and a perfect-gas version of the code has been released to government contractors. Extensive validation runs on blunt and slender bodies with comparisons to experimental data have been completed. Typical run times for a complete case are approximately 4 hours requiring on the order of 4 million words of memory. The graphic shows comparisons to experimental data for heat transfer over a model of the Aeroassist Flight Experiment (AFE) vehicle at Mach 10. Coding of a fully coupled, 2 temperature, 11 species, 23 reaction model for air has been completed. This version will require approximately 30 million words of memory when it becomes fully operational.

Significance:
Present results are being used in the analysis and design of the AFE.

Future Plans:
Complete debugging of the nonequilibrium code and run several points on the AFE trajectory. Effects of changes in the kinetic model will also be studied.

Publications:
AIAA 87-0280.

CFDAER: AEROELASTIC APPLICATIONS FOR AIRCRAFT

Guru P. Guruswamy, Principal Investigator
NASA Ames Research Center

Research Objective:
Objective is to develop a CFD capability to conduct aeroelastic analysis of the complete aircraft. This capability is required for aircraft of national importance for defense considerations like the ATF, F-16, B-1 aircraft; etc. This would provide an opportunity for a "breakthrough" calculation, via a demonstration of the capability to simulate transonic aeroelasticity for improving the performance of the aircraft.

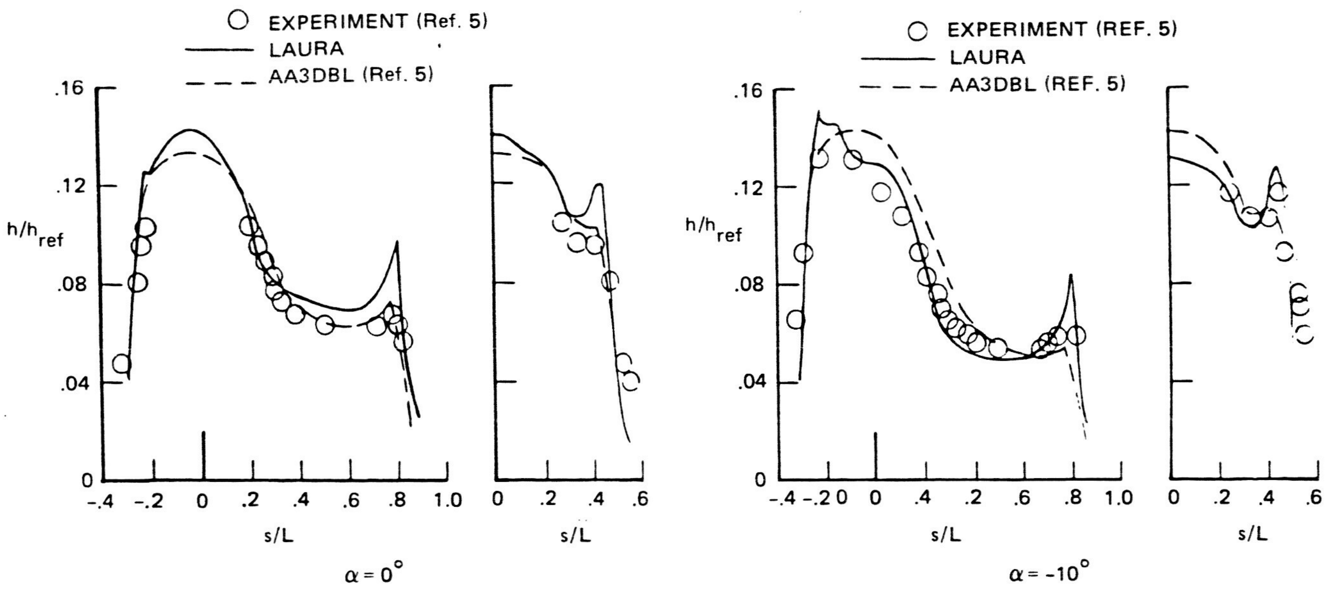
Approach:
Unsteady transonic methods based on the established CFD algorithms will be used. First, a code based on the small disturbance transonic equations will be developed to simulate unsteady flow over the aircraft, including viscous effects. This code will have the capability of simultaneously integrating the aerodynamic and structural equations for coupled calculations. The code being developed uses a finite-difference, alternating-direction implicit (ADI) algorithm for aerodynamic calculations. This technology will be further extended by using the Euler/Navier Stokes equations.

Accomplishment Description:
The present project was selected by NAS as an alternate. Due to the uncertainty of the availability of the Cray-2 time as an alternate project, it was difficult to plan the usage of Cray-2 for this high computer resource demanding project. Typical aeroelastic response analysis for isolated fighter wings require about 3 to 5 hours of Cray-XMP time by using the small disturbance equations on a moderate grid of 70,000 points. For wing body configurations the computational effort required is about 5 times that required for the isolated wings. In typical aeroelastic analysis several response analyses are required. However, effort was made to convert the code to run on Cray-2. One version of the code ATRAN3S is being used by NAS for internal research to validate the new features of NAS.

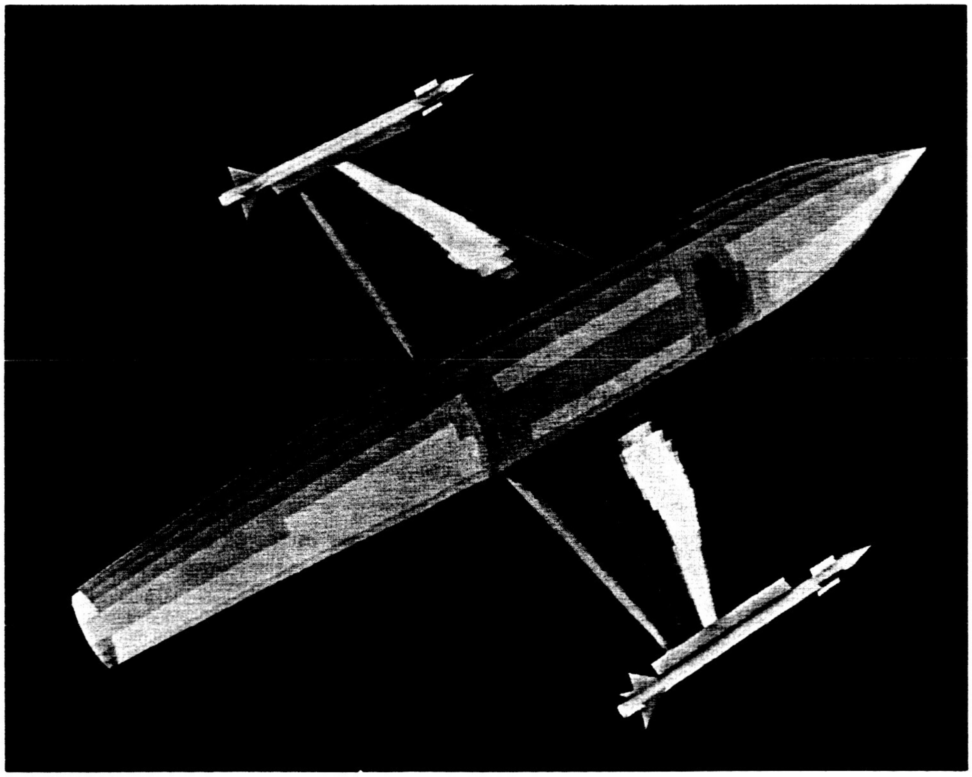
Significance:
This project would provide an opportunity for a "breakthrough" calculation via a demonstration of the capability to simulate transonic aeroelasticity for improving the performance of the aircraft. This development is of great national importance, particularly in view of the Air Force's ATF project.

Future Plans:
This project, which has a heavy demand on computer resources, needs to be completed as soon as possible to provide the technology to industry for the understanding of the aeroelastic phenomenon of complete aircraft, particularly in the transonic regime. It is proposed to complete the first stage of this project by using the small disturbance equations by 1989 and possibly apply for the analysis and design of ATF. The progress depends on the availability of computer and man-power resources. A color photograph showing the preliminary surface pressure results for a typical fighter aircraft obtained on the base code running on Cray-XMP is enclosed with this report. More detailed aeroelastic studies will be conducted on almost complete aircraft when the final code is implemented on Cray-2.

AFE HEATING



GNOFFO: LANGLEY AEROTHERMODYNAMIC UPWIND RELAXATION ALGORITHM



GURUSWAMY: CFDAER: AEROELASTIC APPLICATIONS FOR AIRCRAFT

COMPUTATIONAL STRUCTURAL MECHANICS FOR AEROSPACE PROPULSION AND POWER SYSTEMS

Dale A. Hopkins, Principal Investigator
NASA Lewis Research Center

Research Objective:
Develop integrated structural analysis and optimal design methods for advanced aerospace propulsion and power systems and components.

Approach:
Investigate novel techniques for nonlinear structural analysis and optimization, and develop new computational modules and integrated computer codes leading to the formation of a software testbed to perform and evaluate critical computational investigations of advanced algorithms and structural concepts.

Accomplishment Description:
Two new computer codes, namely MHOST and BEST3D, are currently being developed under Lewis sponsorship. The MHOST code is based on mixed/iterative finite element methodology while the BEST3D code is based on advanced boundary element methodology. These codes represent new technology tools for performing three-dimensional inelastic structural analysis of high temperature energy system structures.

During IIOC the MHOST code was successfully implemented on the NAS Cray-2 and several validation problems were executed. The validation problems involved the static and dynamic structural response analysis of a Space Shuttle Main Engine High Pressure Fuel Turbopump turbine blade.

Significance:
Provides the means for more accurate and efficient computation of the 3-D cyclic nonlinear thermomechanical stress-strain behavior of complex engine structures.

Future Plans:
Continue investigations using the MHOST code and begin implementation of the BEST3D code.

THREE-DIMENSIONAL VISCOUS FLOW AROUND SUBMARINE HULL AND IN THE WAKE

T. T. Huang, Principal Investigator
Co-investigators: Y. T. Lee and S. Ohring
David Taylor Naval Ship R&D Center

Research Objective:
Steady and unsteady flow computations on a turning submarine hull with appendages in the wake.

Approach:
Reynolds-averaged Navier-Stokes equations are solved. For steady flow the computer code INS3D, developed by Dr. Kwak at NASA Ames, is modified for use in the ship hull and wake calculations.

Accomplishment Description:
During the IIOC operational phase, a numerical calculation case for flow past an appendage-flat plate junction was completed. The comparisons between computed results and measured data are good. An average Cray-2 job for this case uses 2-million word memory for approximately half an hour of CPU time.

Significance:
The capability developed from this project will lead to a significant increase in the understanding of ship hydrodynamics and to an improvement in ship design.

Future Plans:
Continuation of this development for more complex geometries is projected. Increased usage of the Cray-2 computer is expected.

NO GRAPHICS SUBMITTED

CONCURRENT NAVIER-STOKES SIMULATIONS

Gary M. Johnson, Principal Investigator
Co-investigators: Daniel V. Pryor, Johnny P. Ziebarth, and Julie M. Swisshelm
Institute for Scientific Computing

Research Objective:
To simulate three-dimensional compressible flows governed by the Navier-Stokes equations with an efficient algorithm well-suited for use on advanced-architecture supercomputers.

Approach:
MacCormack's method with multigrid convergence acceleration to steady state with a zonal equation solver and embedded grid refinements, vectorization, and multitasking.

Accomplishment Description:
An efficient algorithm designed to be used for Navier-Stokes simulations of complex flows over complete configurations has been developed. The algorithm incorporates a number of elements, including an explicit three-dimensional flow solver, embedded mesh refinements, a zonal equation hierarchy ranging from the Euler equations through the full Navier-Stokes equations, multiple-grid convergence acceleration and extensive vectorization and multitasking for efficient execution on parallel-processing supercomputers. Results have been obtained for a preliminary trial of the method on a problem representative of turbomachinery applications. Based on this performance data, it is estimated that a mature implementation of the algorithm will yield overall speedups ranging as high as 100.

Significance:
Potential to reduce simulation times by a factor of 100.

Future Plans:
Further multitasking work on Cray-2 and successor machines.

Publications:
Multitasked Embedded Multigrid for Three-Dimensional Flow Simulation
Multigrid for Parallel-Processing Supercomputers
Development of an Aerodynamics Algorithm for Parallel-Processing Supercomputers, to appear.

LARGE-EDDY SIMULATIONS OF RAMJET COMBUSTION INSTABILITY

Wen-Huei Jou, Principal Investigator
Co-investigator: Suresh Menon
Flow Research Company

Research Objective:
To investigate the flow field and the mechanism of self-sustained oscillations in a ramjet combustor by large-eddy simulations. These oscillations are related to the combustion instability.

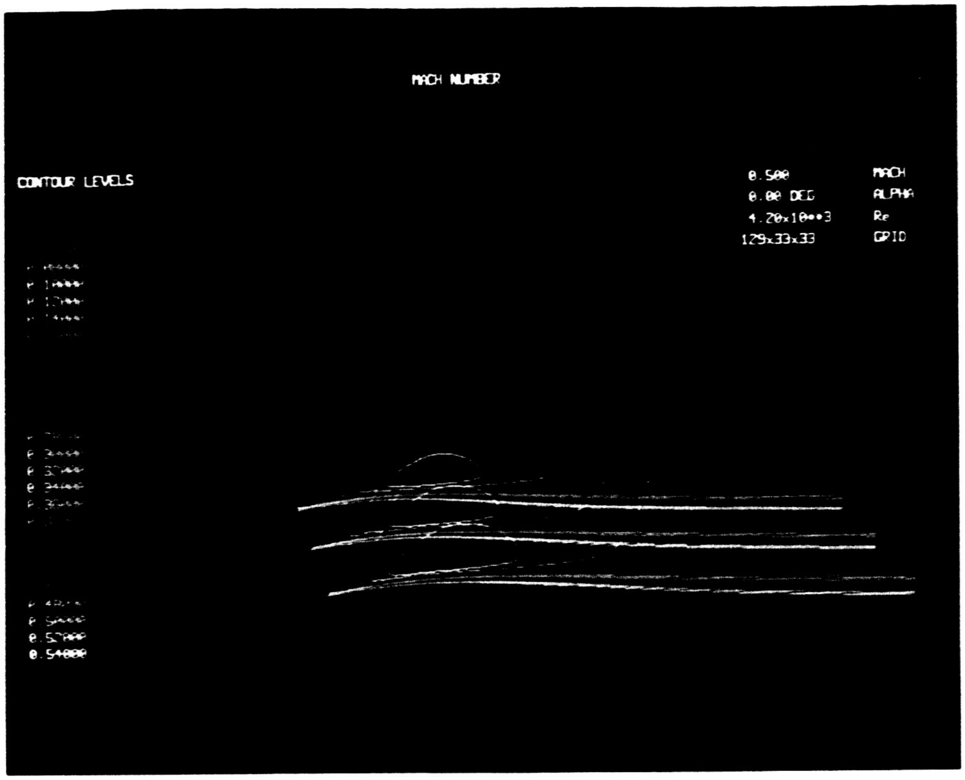
Approach:
An unsteady Navier-Stokes finite volume scheme is used to study acoustic wave and vortex motion in a compressible fluid.

Accomplishment Description:
Two simulations were performed on a grid of 256X64 points which models a ramjet combustor consisting of an inlet duct, a dump combustor and a convergent-divergent nozzle. The inflow Mach number is 0.32, and the flow downstream of the choked nozzle is supersonic. For this grid, a Cray-2 memory of abour 4 MW is required. The phenomenon of self-sustained oscillations in a ramjet combustor requires a real time of $\sim$0.1 s to develop. Although the computer code is efficient, a simulation requires about 18 hours of Cray-2 CPU time to obtain sufficient data for meaningful statistical analysis of the flow field. The formation of large-scale coherent structures due to the rollup and merging processes in the separated shear layer is observed. The instantaneous dilatation field contours represent the near-field acoustic signature. Around each isolated vortex, the instantaneous dilatation field behaves as a quadrupole. Near the location where large-scale structures impinge on the wall, a multipole expansion of the instantaneous dilatation field gives a weak monopole and a strong dipole, and thus this region is identified as a potential source of high acoustic radiation. The vorticity spectrum at the impingement region was compared to the pressure spectra at the base of the step where vorticity is insignificant. The comparison reveals that two types of self-sustained oscillations are possible. One is the acoustic resonant oscillation and the other is the acoustic-vortex coupled-mode oscillation. A physical model for the coupled-mode oscillations based on the results of numerical simulations was constructed to explain the physical phenomena.

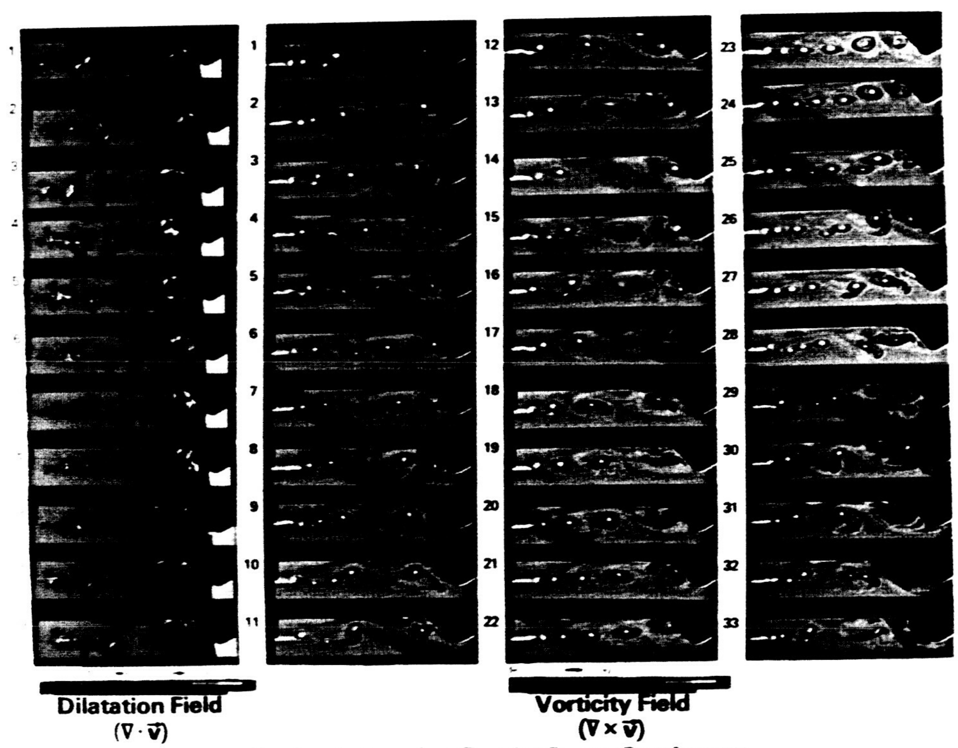
Significance:
Understanding the mechanism of self-sustained oscillations in a ramjet combustor is an essential step toward the control of the combustion instability observed in a ramjet.

Future Plans:
Simulations of cases with combustion are planned for further investigation of combustion instability in a ramjet.

Publications:
Two papers in preparation: Simulations of Ramjet Combustor Flow Field – Part I. Numerical Model, The Mean and Large-Scale Motion, AIAA Paper 87-1421, and Part II. The Origins of Pressure Oscillations, AIAA Paper 87-1422.



JOHNSON: CONCURRENT NAVIER-STOKES SIMULATIONS



Unsteady Flow in a Ramjet Dump Combustor

JOU: LARGE-EDDY SIMULATIONS OF RAMJET COMBUSTION INSTABILITY

NUMERICAL SIMULATIONS OF UNSTEADY, COMPRESSIBLE SPATIALLY EVOLVING SHEAR FLOWS

K. Kailasanath and F. F. Grinstein, Principal Investigators
Co-investigators: E. S. Oran and J. P. Boris
Naval Research Laboratory

Research Objective:
To study the fundamental mechanisms involved in the spatial and temporal evolution of large scale structures in (a) reactive flow in an axisymmetric ramjet and (b) nonreactive, 3-D planar mixing layer.

Approach:
To solve the time-dependent, compressible flow equations. The chemical-kinetic terms are coupled using time-step splitting techniques.

Accomplishment Description:
a) In the ramjet simulations, a mixture of hydrogen and air flows through an axisymmetric inlet into a combustor of larger diameter and then exits through a choked nozzle. The reactive flow field in the combustor is found to be dominated by large-scale coherent vortex structures. The dynamics of these structures is significantly influenced by the acoustics of the inlet and the combustor. Investigations of the effects of fuel-air ratio and energy release parameters show that under certain circumstances flame extinguishment can occur due to effects of flame stretch induced by vortices. An instantaneous snapshot of the flow field showing one such occurrence is presented in Fig. 1. About 24 hours of CPU time were used, with an average memory requirement of about 2.5 million words.

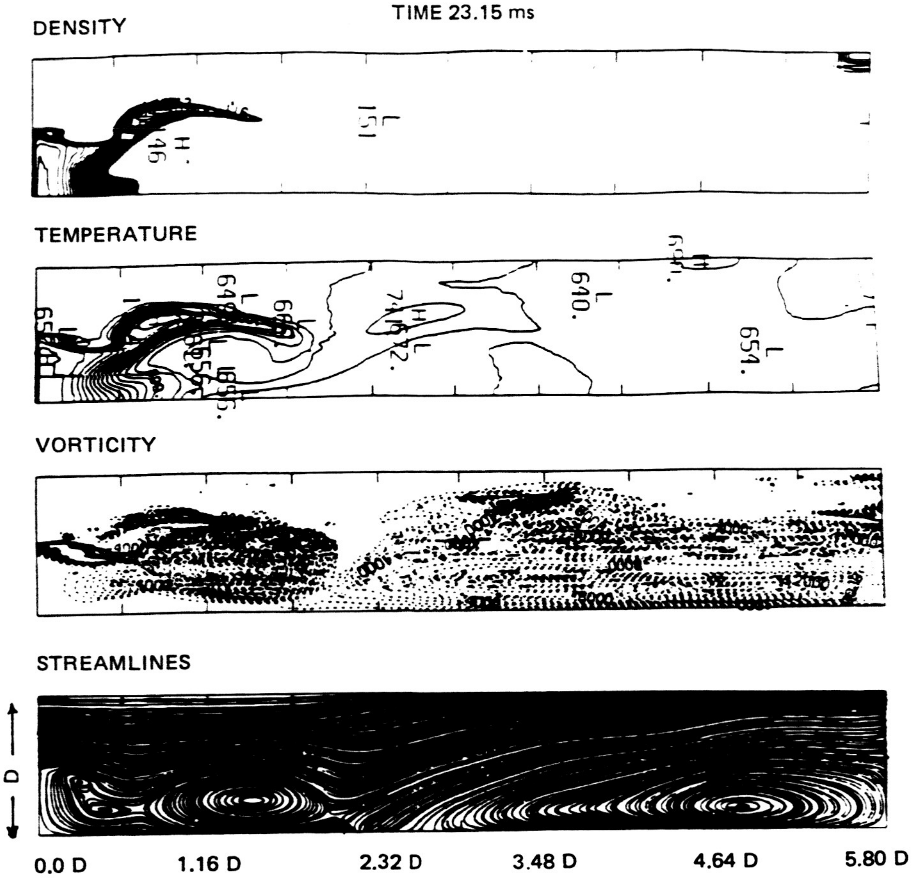
b) In the 3-D mixing layer simulations, secondary instabilities were triggered by introducing a spanwise sinusoidal perturbation of the cross-stream momentum of an initially two-dimensional mixing layer. The surfaces in Figs. 2a and 2b enclose the regions where the vorticity has more than 15% of its peak value. The mean Mach number is about 0.3, and the free-stream velocity ratio is 10:1. Streamwise vortices (ribs) of opposite signs appear in pairs in the region of the braids. They coalesce at the level of the two-dimensional spanwise roll-ups, where they either reduce or enhance locally the spanwise vorticity. About 16 hours of Cray-2 CPU time were used, with average memory requirements of 4 million words per job.

Significance:
a) The reactive flow simulations are leading to a better understanding of combustion instabilities in ramjets and also the effects of flame stretch in turbulent shear flows.

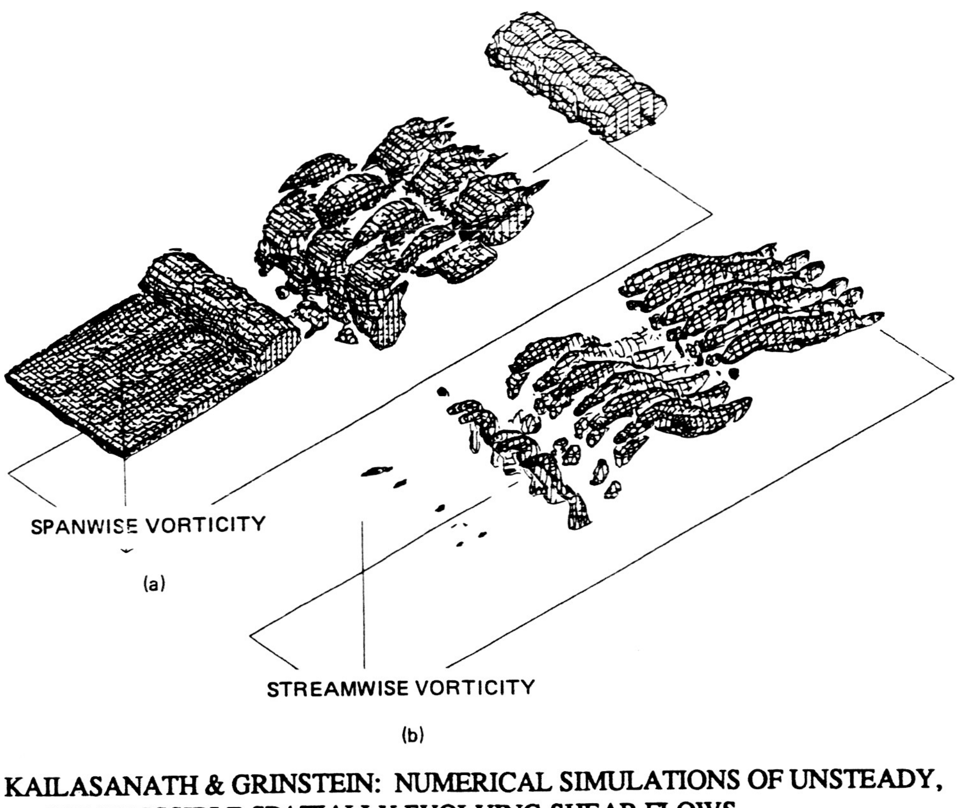
b) The 3-D simulations help understand the role of coherent structures on mixing and the transition to turbulence of shear flows.

Future Plans:
a) To study the 3-D, reactive shear flows.

b) To investigate the effect of unsteady initial conditions on the 3-D instabilities of the mixing layer, and study the interaction of two such mixing layers in a planar jet flow configuration.



KAILASANATH & GRINSTEIN: NUMERICAL SIMULATIONS OF UNSTEADY, COMPRESSIBLE SPATIALLY EVOLVING SHEAR FLOWS



KAILASANATH & GRINSTEIN: NUMERICAL SIMULATIONS OF UNSTEADY, COMPRESSIBLE SPATIALLY EVOLVING SHEAR FLOWS

EULER ANALYSIS OF A COMPLETE F-16

Steve L. Karman, Jr., Principal Investigator
Co-investigators: Christopher Fouts, John Chawner, Greg Spragle, Louis Hunter, Richard Matus, John Steinbrenner, Mike Remotigue, and George Howell
General Dynamics

Research Objective:
To develop and validate an Euler analysis capability for external flow fields.

Approach:
Analyze complex external aircraft and components with multiple blocked CFD codes and compare with available experimental data. The F-16 geometry serves as the test case for a complete aircraft configuration. An isolated F-16 wing will be analyzed in a grid refinement study.

Significance:
The complete F-16 configuration (including wing, body, horizontal and vertical tail, inlet, nozzle and ventral fin) was successfully analyzed at one transonic flow condition.

Future Plans:
Analyze an isolated wing to evaluate the shock capturing capability of the CFD code. The F-16 analysis will then continue at additional flow conditions.

Publications:
S. L. Karman, Jr., J. P. Steinbrenner, K. M. Kisielewski: Analysis of the F-16 Flow Field by a Block Grid Euler Approach. Paper No. 18, 58th Meeting of the Fluid Dynamics Panel Symposium on Applications of Computational Fluid Dynamics in Aeronautics, April 1986.

TRANSONIC NAVIER STOKES WING PROJECT

Principal Investigator: Unver Kaynak
Co-investigator: Jolen Flores
NASA Ames Research Center

Research Objective:
Develop computational methods to numerically simulate transonic separated flows around wings both in free-air and in wind tunnels.

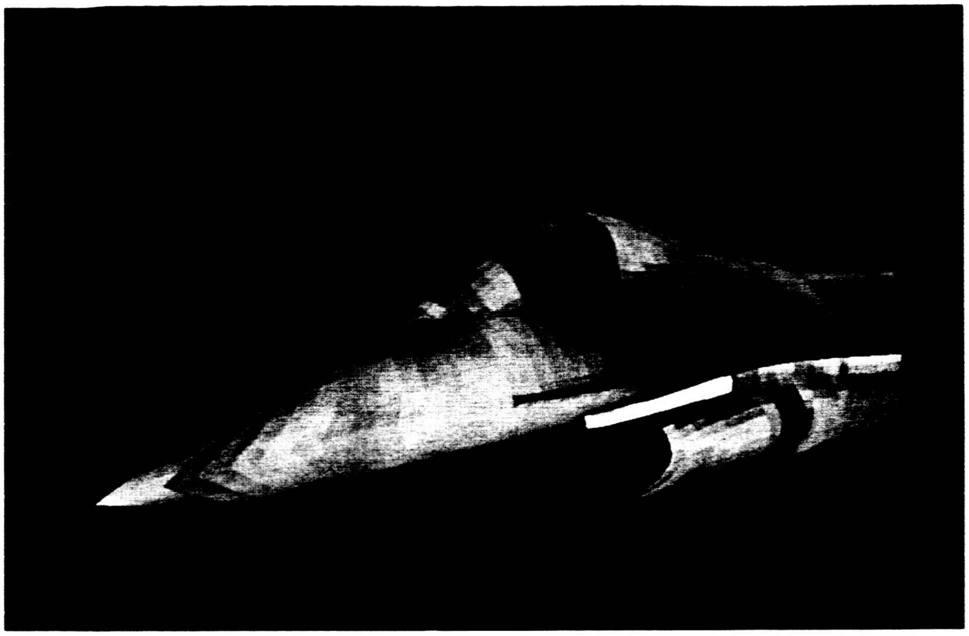
Approach:
The Transonic Navier-Stokes (TNS) program, which solves the Euler/Navier-Stokes equations in a zonal fashion, is used.

Accomplishment Description:
A series of transonic separated flows using different sized grids were numerically simulated around a swept wing. Data is available from an experiment conducted at NASA Ames High Reynolds Number Channel I. As a technology demonstration, one solution with extremely fine grid resolution (1.1 million total grid points) using the Cray-2 supercomputer was achieved. Of particular importance for flow physics, by using a moderate size grid, a transonic mushroom-type separated flow with two counter-rotating vortices was demonstrated for the first time, which closely resembles the experimental pattern. Depending on each case, Cray-2 memories ranging from 15 to 42 megawords, and run times ranging from 1 to 90 hours were used.

Significance:
The computer program (TNS Wing Version), and the results, are directly usable by the aeronautical society for wind-tunnel simulations. This method could lead to cut-down in wind-tunnel costs, and could be used for designing transonic wings, as well as designing new experiments.

Future Plans:
New turbulence models with nonequilibrium effects are being investigated.

Publications:
Kaynak, Ü. and Flores, J.: Advances in the Computation of Transonic Separated Flows Over Finite Wings. AIAA Paper 87-1195, AIAA 19th Fluid Dynamics, Plasma Dynamics and Lasers Conference, Honolulu, Hawaii, June 8-10, 1987.



KARMAN: EULER ANALYSIS OF A COMPLETE F-16

KAYNAK: TRANSONIC NAVIER-STOKES WING PROJECT

ANALYSIS OF COMPRESSIBLE FREE SHEAR LAYER INSTABILITIES IN JETS OF ARBITRARY CROSS SECTION

Dr. Shozo Koshigoe, Principal Investigator
Co-investigator: Dr. Nagi N. Mansour
Naval Weapons Center/NASA Ames Research Center

Research Objective:
The aim of this work is to develop a calculational method for analysis of the shear layer instabilities in jets of arbitrary mean flow profiles in order to understand and practically control the flow structures.

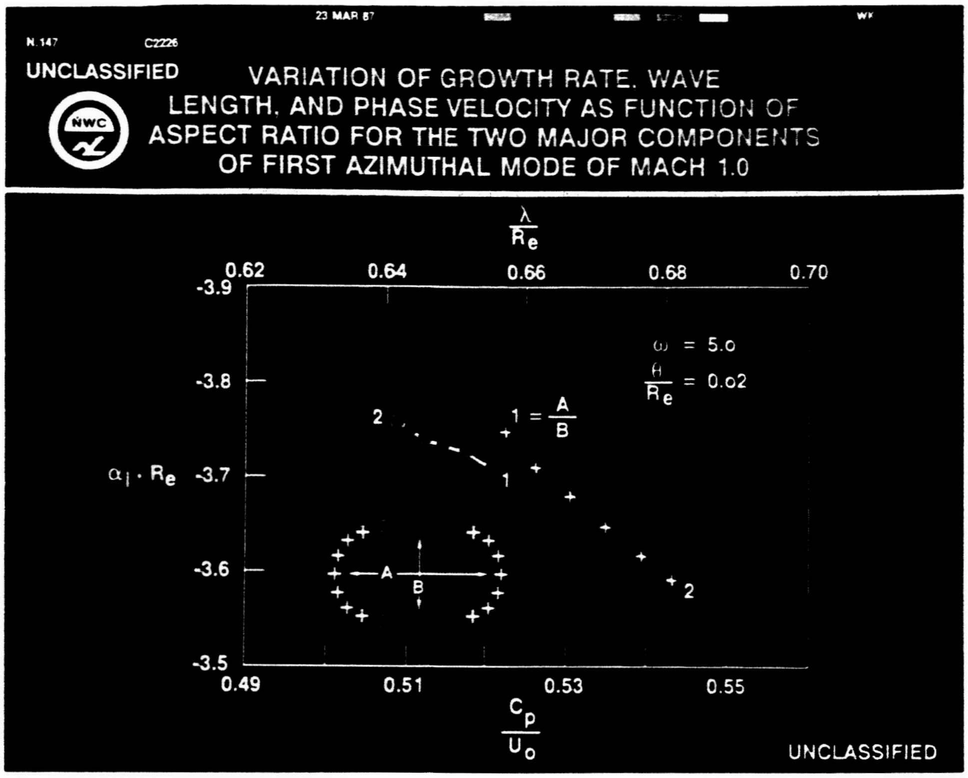
Approach:
A new hybrid form based on both differential and integral equation was applied for the three-dimensional linear stability analysis of jets of arbitrary mean flow profile.

Accomplishment Description:
Green function techniques were used to derive integral forms of two-dimensional Rayleigh equation.[1] The integral equation approach was further improved by the introduction of a new hybrid form of linear stability theory based on both differential and integral equation.[2] They were applied for the stability analysis of elliptic jets.[3] The analysis results indicated that two eigen modes were responsible for early development of the axis switching (bending of the jets). The basic mechanism for bending of the jet was found to be insensitive approximately to Mach number less than 1.5. The number of Cray-2 hours used for this study was approximately 20, and the peak run time memory was 13 MW.

Significance:
The new hybrid scheme provides a means for investigating behaviors of noncircular jets which have important potential on practical applications.

Future Plans:
The method is being applied to jets with other geometries (e.g., triangular,[4] and rectangular). The problems of understanding the relation between instability waves and geometry of jets will be considered as inverse problems as well as forward problems.

Publications:
[1] S. Koshigoe and A. Tubis, Phys. Fluids 29, 3982 (1986).
[2] S. Koshigoe and A. Tubis. To appear in Phys Fluids (June 1987).
[3] S. Koshigoe, A. Tubis, and C. M. Ho. Submitted to Phys. Fluid.
[4] S. Koshigoe, A. Tubis, and N. N. Mansour. To be submitted to Phys. Fluids.



KOSHIGOE: ANALYSIS OF COMPRESSIBLE FREE SHEAR LAYER INSTABILITIES IN JETS OF ARBITRARY CROSS SECTION

MULTIPLE MODULE SCRAMJET INLET ANALYSIS

Ajay Kumar, Principal Investigator
NASA Langley Research Center

Research Objective:
To study high-speed propulsion/airframe integration problem.

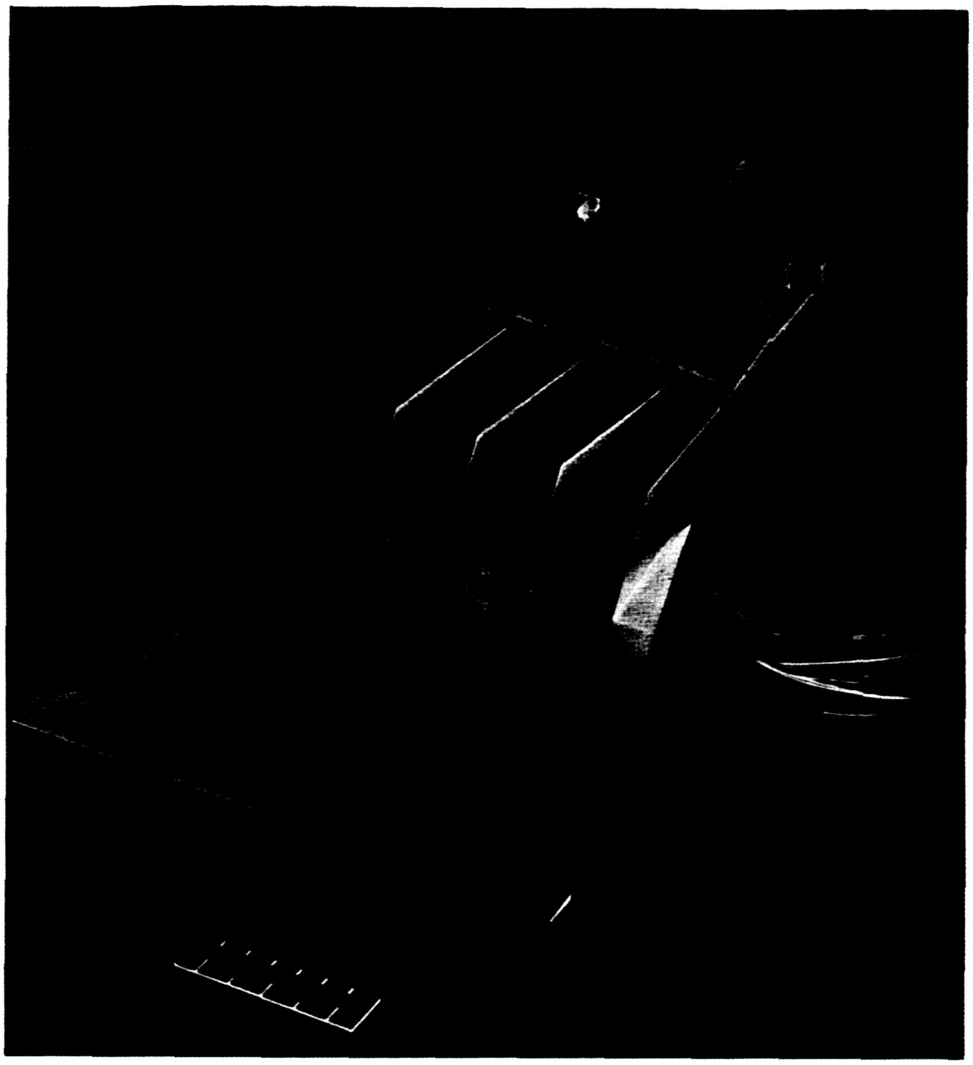
Approach:
An experimental, as well as analytical, program has been devised to study the propulsion/airframe integration problem. The experimental model has three modules mounted on a flat plate (see fig. 1) that simulates the forebody boundary-layer. The model will be tested over a range of Mach numbers, angles of attack, and yaw. The experimental conditions will be numerically simulated using a three-dimensional Navier-Stokes code, SCRAMIN. The results from these calculations will be compared against the experiment.

Accomplishment Description:
Although no experimental results are yet available, some preliminary calculations have been made at zero and small angles of attack and yaw using the geometry of the experimental model and the tunnel flow conditions. For the angle of attack case, only half of the configuration is analyzed due to the flow symmetry whereas at angle of yaw, the whole configuration is analyzed. A total of 330,000 grid points are used for half of the configuration requiring approximately 7 million words storage. A two-dimensional Navier-Stokes code is used on the front flat plate to determine the flow profiles at the face of the modules. These profiles are then used as the inflow conditions for the three-dimensional Navier-Stokes code to solve the flow in the modules. Sample results are presented in figure 2 which shows the pressure contours and velocity vectors in the vertical symmetry plane of one of the modules. The pressure contours show a swept shock structure caused by the swept compression surfaces of the modules. The velocity vector plot shows a significant downturn in flow direction ahead of the cowl resulting in some flow spillage. This downturn in the flow is caused by the sidewall sweep and the interaction between the high-pressure internal flow and low-pressure external flow. Once the inlet flow passes behind the cowl leading edge, it is turned back parallel to the cowl plane. This turning results in a cowl shock on the upper side of the cowl and an expansion on the underside of the cowl which can be seen clearly in the pressure contour plot. The preceeding calculation took approximately 12 hours on Cray-2.

Significance:
The significance of this work is in the study of the performance of the scramjet engine inlets and their interactions with each other and with the vehicle. Comparison with experiments will validate the code for the current flow situation and conditions. The code can then be used to do a parametric study to help design the engine modules.

Future Plans:
The numerical results will be compared with the experimental results as soon as they are available More realistic configurations will then be calculated.

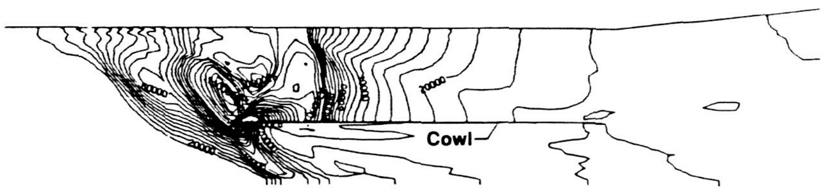
Publications:
Dwoyer, D. L. and Kumar, Ajay: Computational Analysis of Hypersonic Airbreathing Aircraft Flow Fields. AIAA Paper No. 87-0279, Jan. 1987.



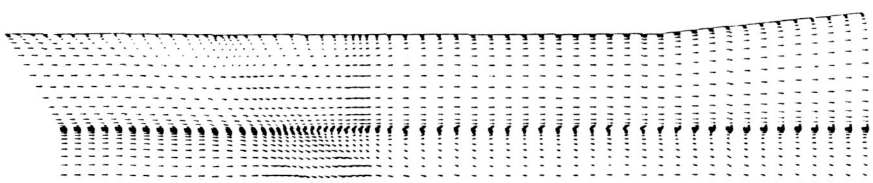
KUMAR: MULTIPLE MODULE SCRAMJET INLET ANALYSIS

PRESSURE CONTOURS AND VELOCITY VECTOR FIELD IN THE SYMMETRY PLANE OF MULTIPLE MODULE SCRAMJET ENGINE

(M_{∞} = 4.03; Grid: 61 x 91 x 61; Geometric Contraction Ratio = 4.0; α - 0°)



Pressure



Velocity Vector Field

KUMAR: MULTIPLE MODULE SCRAMJET INLET ANALYSIS

INCOMPRESSIBLE NAVIER-STOKES/SPACE SHUTTLE MAIN ENGINE

Dochan Kwak, Principal Investigator
Co-investigators: Stuart Rogers and James Chang
NASA Ames Research Center/Rocketdyne

Research Objective:
To simulate flow in the Space Shuttle main engine power head.

Approach:
Use incompressible Navier-Stokes flow solver in generalized coordinates (INS3D code).

Accomplishment Description:
To study the flow around liquid oxygen posts in the SSME power head, multiple posts problems have been simulated during the IIOC period. Approximately 300 hours of Cray-2 was used and about 5 million words of memory was required for an average job.

Significance:
Demonstrated that the complex flow field in the SSME can be simulated.

Future Plan:
Continue CFD analysis of the current and future SSME configurations.

Publications:
AIAA 86-0353.

COMPUTER SIMULATION OF BIOLOGICAL MOLECULES IN THEIR NATURAL ENVIRONMENT

Robert D. MacElroy, Principal Investigator
Co-investigators: Andrew Pohorille and Brian Owenson
NASA Ames Research Center

Research Objectives:
The objective of this research is to provide a molecular level explanation of processes of biological and proto-biological interest. In particular, we are interested in the transport of ions across water/membrane interfaces.

Approach:
Our approach is to simulate the behavior of all the atoms in a system of interest by numerical integration of Newton's equations of motion (molecular dynamics) and by Metropolis Monte Carlo methods. Both methods allow us to calculate thermodynamic and structural properties of interest. Molecular dynamics also allows calculation of time-dependent features.

Accomplishment Description:
1. A water liquid/vapor interface was simulated. The orientations of water molecules at a free surface were found to be different from orientations in the bulk liquid. The mean electric field across the interface was calculated. This project used 30 CPU hours. Average memory use was 1 Mword.

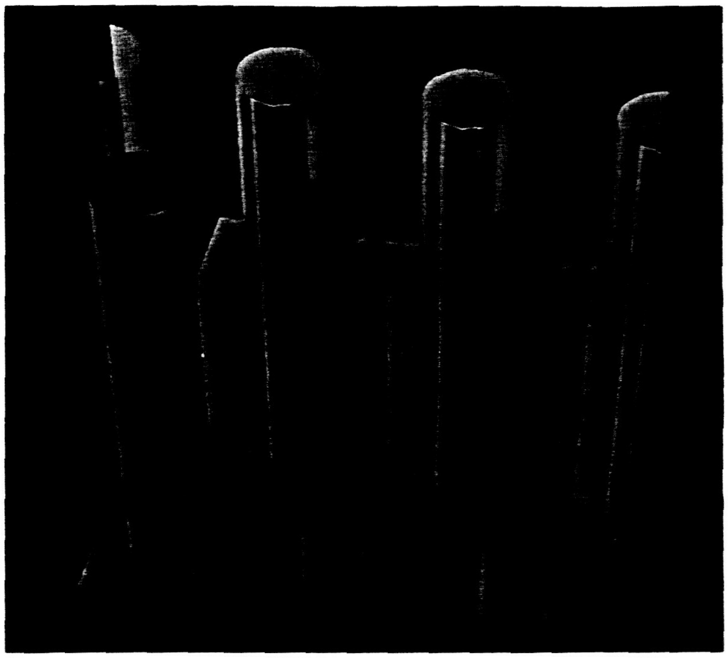
2. Ion capture from water by an ion trapping molecule was simulated. The dynamics of the ion dehydration and capture was explained in molecular detail. It was found that the capturing molecule must contain groups of atoms which interact favorably with the ion. These groups must have structural flexibility in order to assist moving the ion across an energetically unfavorable region. This project used 40 CPU hours. Average memory use was 1 Mword.

3. A new, efficient method for calculating entropies and free energies of aqueous solutions is being developed. An essential assumption of this method is that the structure of rapidly frozen water is similar to the structure of liquid water and therefore can be used as a reference state. This assumption was verified in a series of simulations of frozen aqueous glasses. This project used 30 CPU hours. Average memory use was 8 Mwords.

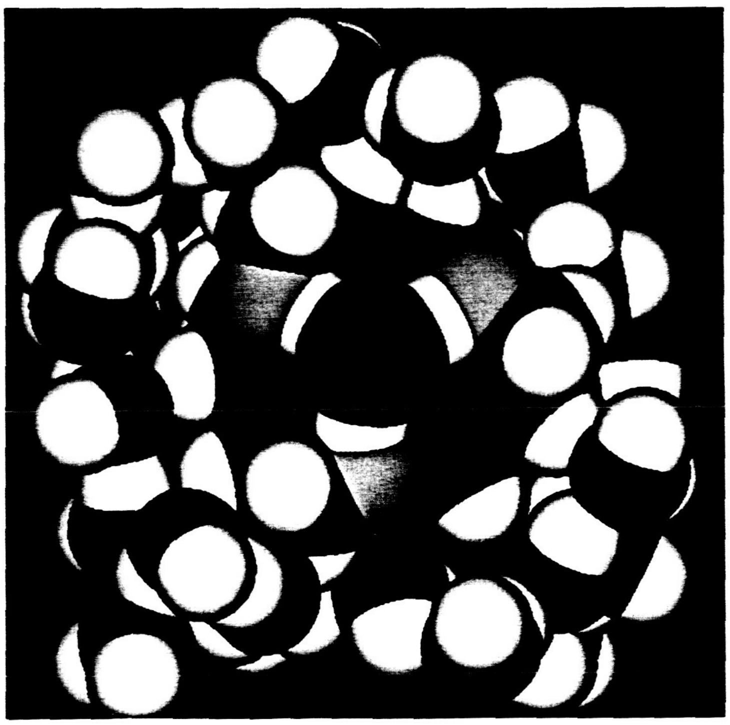
Significance:
These projects were the first state-of-the-art, large scale simulations of their type. The results shed light on the behavior of molecular interfaces and on the principles that determine transport phenomena across membranes. In particular, we found the structural requirements for a trapping molecule to capture an ion from an aqueous environment.

Future Plans:
We intend to continue using computer simulations 1) to explain stabilities of membranes made of various types of amphiphilic molecules, 2) to study transport of ions across membranes, and 3) to complete the development of the method of calculating entropies and free energies of aqueous solutions.

Publications:
1. M. A. Wilson, A. Pohorille, and L. R. Pratt: Molecular Dynamics of the Water Liquid-Vapor Interface. J. Chem. Phys., to appear in May 1987.
2. A. Pohorille, L. R. Pratt, R. A. LaViolette, M. A. Wilson, and R. D. MacElroy: Comparison of the Structure of Harmonic Aqueous Glasses and Liquid Water. J. Chem. Phys., in press.



KWAK: INCOMPRESSIBLE NAVIER-STOKES / SPACE SHUTTLE MAIN ENGINE



Chloride ion $C\ell^-$ (green) captured by an ion-trapping molecule, cryptand SC24. Nitrogen atoms are blue, oxygen atoms are red, carbon atoms are black, and hydrogen atoms are white. The cryptand is surrounded by a layer of water molecules (red and white). Charged and flexible $N - H^+$ groups of the cryptand (blue and white) that interact directly with the ion provide the assistance necessary to capture the ion from the aqueous environment.

MACELROY: COMPUTER SIMULATION OF BIOLOGICAL MOLECULES IN THEIR NATURAL ENVIRONMENT

TRANSONIC FLOW ABOUT THE F-16A AND OTHER COMPLEX CONFIGURATIONS

Michael D. Madson, Principal Investigator
Co-investigators: Alex C. Woo and Ralph L. Carmichael
NASA Ames Research Center

Research Objective.
To develop and validate a computational method which eliminates the use of surface-conforming grids in the analysis of complex aircraft configurations in the transonic flow regime.

Approach:
The *TranAir* full-potential code, being developed by Boeing under NASA contract, embeds accurately defined surface panel models of configurations such as the F-16A in a rectangular array of flow field grid points.

Accomplishment Description:
The *TranAir* code was initially developed on a Cray X-MP/48, and was converted to run on the Cray-2 upon its arrival. Conversion to the Cray-2 involved some modifications to the original code due to differences in the two environments. The large amount of available memory on the Cray-2 led to two significant enhancements to the code. The first was dynamic array storage allocation utilizing a memory manager, dramatically reducing the dependence of the code on problem size. A single parameter is defined at the beginning of the program which identifies the size of the heap that will contain all arrays which are problem-size dependent. The second enhancement was the implementation of core-resident I/O, which emulates the fast access storage capability of the SSD on the Cray X-MP/48. While validation of the Cray-2 version of *TranAir* is not complete, some complete cases have been run. As of February 1987 a total of 28 CPU hours had been used in the conversion and initial validation of the code. Current solutions require about 2 MW of memory, although these solutions are preliminary, and a substantially larger amount of memory will be utilized for future work.

Significance:
The *TranAir* transonic solution about the complete F-16A geometry represents a major advancement in the complexity of aircraft configurations for which transonic solutions may be computed.

Future Plans:
Plans call for the completion of the validation process, at which time solutions utilizing 2 to 4 million grid points will be generated. Configurations will include fully defined aircraft with external stores and deflected surfaces such as flaps and leading edge slats.

Publications:
Everson, B. L., Bussoletti, J. E., Johnson, F. T., Samant, S. S., Erickson, L. L., and Madson, M. D.: TranAir and its NAS Implementation. Presented at the NAS Conference on "Supercomputing in Aerospace," March 1987.

AIRLOADS AND ACOUSTICS OF ROTORCRAFT

W. J. McCroskey, Principal Investigator
Co-investigators: J. D. Baeder, C. L. Chen, and G. R. Srinivasan
NASA Ames Research Center

Research Objective:
Develop and validate viscous CFD codes for three-dimensional, unsteady aerodynamic flows about arbitrary rotorcraft configurations, including isolated rotors and rotor-body combinations.

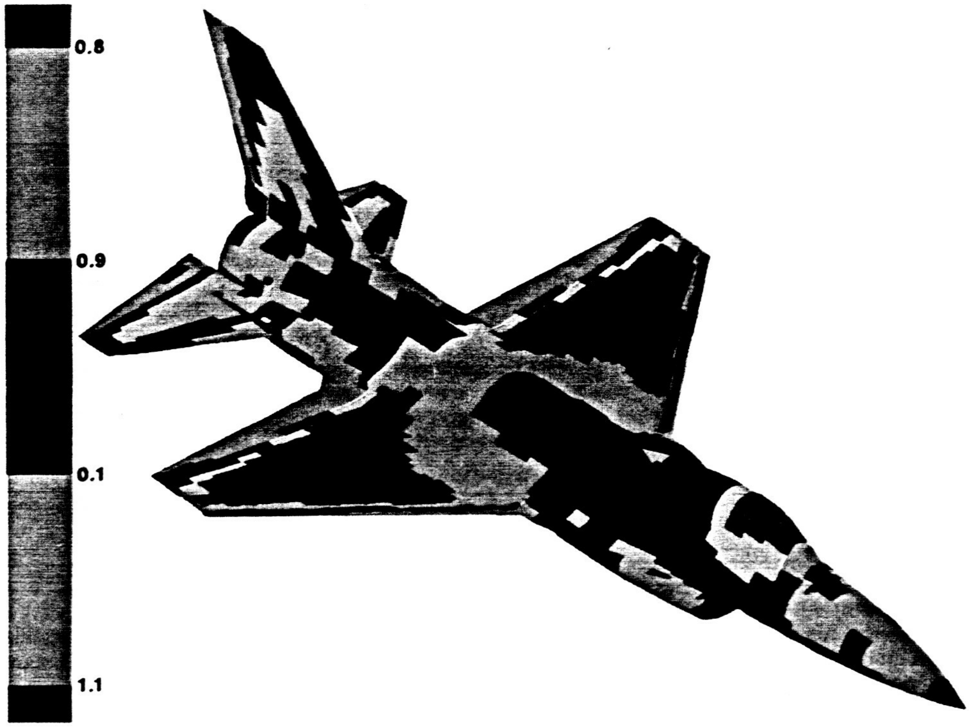
Approach:
Adapt and extend unsteady thin-layer Navier-Stokes equations to rotorcraft configurations. Couple flow solvers with zonal grid topologies, including rotating and nonrotating blocks. Develop special grid clustering and wave-fitting techniques to capture low-level radiating acoustic waves.

Accomplishment Description:
The formation and initial rollup of tip vortices have been accurately calculated, analyzed, and compared with measurements on nonrotating blade tips. The attached photograph shows the initial vortex formation for two tip geometries. Preliminary Euler solutions have been obtained on rotors with multiple blades, and the correct bounds of aeroacoustic "delocalization" have been demonstrated. A fifth-order accurate Navier-Stokes scheme has been used to compute acoustic waves in two-dimensional studies, where previous methods had shown limitations away from the body.

Significance:
The understanding and accurate prediction of tip vortices, acoustic wave propagation, and rotor-body interactions are essential to the improvement of rotorcraft.

Future Plans:
Hover and forward-flight calculations are in progress. Viscous effects on more advanced and complex tip geometries will be studied. A rotor-body code will be developed and validated.

Publications:
AIAA 86-1095.



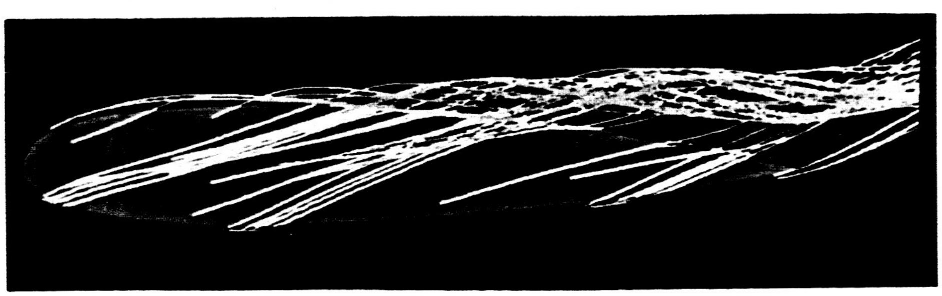
TranAir local mach no. results for the F-16A.
$M_\infty = 0.9$, $\alpha = 4^\circ$

MADSON: TRANSONIC FLOW ABOUT THE F-16A AND OTHER COMPLEX CONFIGURATIONS

TIP VORTEX FORMATION — RECTANGULAR WING

$M_\infty = 0.17$, $\alpha = 11.8^\circ$, Re = 2 million

(a) ROUND TIP



(b) SQUARE TIP

MCCROSKEY: AIRLOADS AND ACOUSTICS OF ROTORCRAFT

DETERMINATION OF FLOW FIELD AROUND AN OBLIQUE-WING USING THE NAVIER-STOKES EQUATIONS

U. B. Mehta, Principal Investigator
Co-investigator: M. Yarrow
NASA Ames Research Center

Research Objective:
Determination of flow field around an oblique-wing aircraft using the chimera-multiple embedded grid method to solve Navier-Stokes equations for full wing-body-control surfaces configuration.

Approach:
Chimera-multiple embedded grid method is used to model the wing-body configuration and then the full wing-body-control surfaces configuration. The Navier-Stokes equations are solved on separate grids, these solves being multitasked as separate concurrent processes, the results of which are updated on overlapping grid regions by interpolation.

Accomplishment Description:
The flow past an oblique-wing using the thin-layer Navier-Stokes equations was computed. The thin-shear-layer approximation and algebraic eddy-viscosity turbulence model were used to simplify the Reynolds-averaged Navier-Stokes equations. An implicit, factored numerical scheme and concept of pencil data structure were utilized. Grid refinement studies were conducted. Interactive Cray-2-Iris graphics tools were utilized to convert the computed flow data to animations of the flow field about the oblique-wing.

Test programs (simple elliptic PDE solved by multitasked zonal finite-differences) were designed to test the Cray-2 Fortran Multitasking Library and determine its applicability to the Chimera method. The multitasking library was successfully called by address from a "C" main program, which was a requirement of this Chimera implementation.

Significance:
Multitasking will provide the next improvement in computational speed needed by more complex, larger and more capable methods such as Chimera. The Multitasking Multiple-Embedded-Grid Solver currently being developed and tested by these investigators is remarkable in several ways: 1) It multitasks, thereby completing sooner and freeing the system for use by other users; 2) a "C" main program uses Fortran multitasking library calls to spawn multiple flow solvers; 3) large amounts of memory are effectively managed in "C" and given to Fortran routines; computation occurs in Fortran to exploit vectorization.

Future Plans:
Completion of the Multitasking Multiple-Embedded-Grid-Solver will permit whole-body aerodynamic simulation of the Oblique-Wing Research Aircraft. The code will also be used as a platform with which to perform important experiments in multiple-grid zonal and embedded methods and multitasking of solvers.

STABILITY AND ACTIVE CONTROL OF BOUNDARY LAYER FLOWS

Paresh Parikh, Principal Investigator
Co-investigators: Lucio Maestrello and Alvin Bayliss
NASA Langley Research Center/Northwestern University

Research Objective:
To study spatial stability of boundary layer flows, active control of unstable disturbances by localized surface heating and the resulting sound radiations.

Approach:
Three-dimensional compressible, Navier-Stokes equations are used to calculate growth of unstable disturbances in a boundary layer. As a means to delay transition, growth of these disturbances is inhibited by introducing a temperature signal from the surface which produces an out-of-phase component, thereby reducing the disturbance level.

Sound radiation in the far-field is calculated separately by solving convective wave equation and using unsteady pressure on the surface calculated from the Navier-Stokes solution, as a time dependent boundary condition. Effect of active control on sound radiation is also studied.

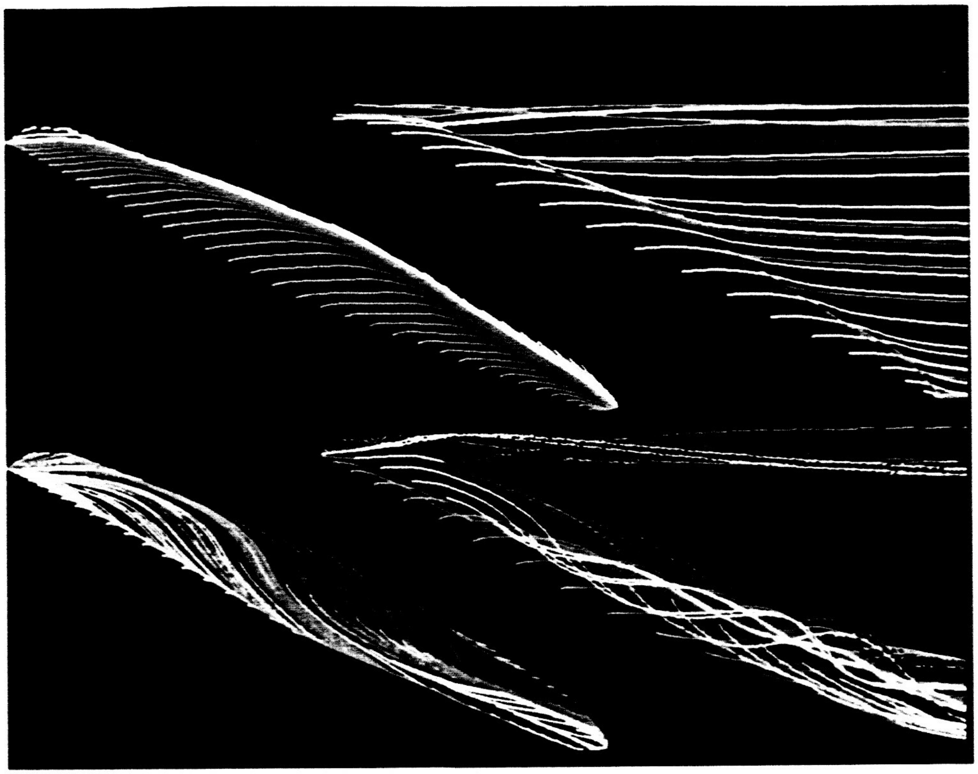
Accomplishment Description:
During the initial period ending February 28, 1987, only the convective wave equation program, described above, was run on the NAS. Other programs could not be run on the NAS as they were initially written for Cyber 205 in semicolon vectorized notations. The program conversion is under way currently.

Sound radiation was calculated for a wave-packet instability disturbance convecting over a concave-convex surface. Sound radiation was again calculated when active control was applied.

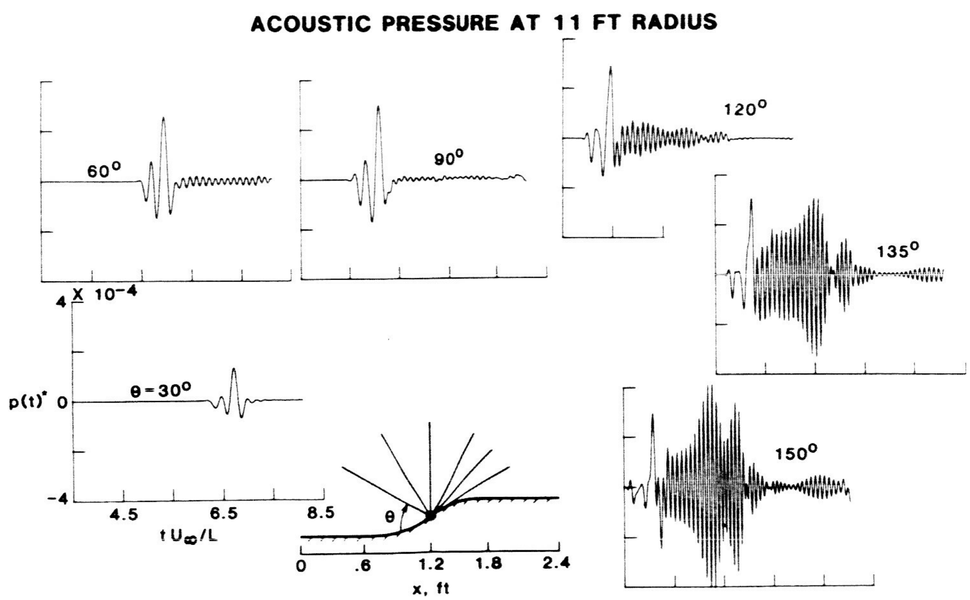
Significance:
It is very important to understand the effect of stability and active control on the resulting sound. While the application of active control may result in a stable boundary layer, the resulting sound may be detrimental to the flow downstream. A wind tunnel contraction is a good example, where an attempt to maintain a laminar boundary layer on the contraction walls using active control may have an adverse effect on the model downstream.

Future Plans:
Since stability study on a three-dimensional configuration calls for resolving the fine structure in a boundary layer, a large grid is needed. For this purpose NAS will be used. First, the available vectorized codes will be converted for efficient operation on Cray. Then study of boundary layer stability and active control on three-dimensional configurations will be undertaken.

Publications:
Based on the above work, the following paper was presented in the NASA NLF and LFC Symposium held at NASA LaRC during March 16–19, 1987. Title: Application of Sound and Temperature to Control Boundary Layer Transition. Authors: L. Maestrello, P. Parikh, A. Bayliss, L. S. Huang, and T. D. Brynt.



MEHTA: DETERMINATION OF FLOW FIELD AROUND AN OBLIQUE-WING AIRCRAFT USING THE NAVIER-STOKES EQUATIONS



PARIKH: STABILITY AND ACTIVE CONTROL OF BOUNDARY LAYER FLOWS

UNDERSTANDING THE EFFECTS OF INITIAL CONDITIONS ON TROPOPAUSE FOLDS

Leonhard Pfister, Principal Investigator
NASA Ames Research Center

Research Objective:
To understand the fluid dynamical processes governing tropopause folding events associated with midlatitude storms.

Approach:
Numerical simulation of both realistic and idealized cases of tropopause folds.

Progress:
During the NAS IIOC we modified a mesoscale meteorological simulation model for midlatitude weather systems, running on the Ames Cray-XMP, for use on the NAS Cray-2. This involved changing from UPDATE to the Source Code Control System and some major changes in the printer output section of the code. We also took this opportunity to vectorize sections of the code which had not been vectorized in the XMP version of the code.

In a similar way, we have modified the Cray-XMP code which initializes the mesoscale model for idealized analytic initial conditions for use on the Cray-2. For the simulation code, we obtained results identical to those on the XMP on the Cray-2 to six significant figures after 48 hours (~960 time steps) of integration time. The lengthy initialization code also produced identical results on both machines. Thus, we are satisfied that we have successfully implemented our codes on the Cray-2. Unfortunately the IIOC ended before we could begin real production runs. Thus, the enclosed figures represent results from Cray-2 runs that were also obtained on the XMP. I would estimate that we used less than 1 hour of CPU time to revise and debug our codes.

Significance and Future Plans:
If we can obtain some Cray-2 time for the IOC, we expect to make production runs. The finer resolution obtainable with the large Cray-2 memory will enable us to resolve the tropopause folds realistically (which is not possible on the XMP).

NUMERICAL RESEARCH OF TWO- AND THREE-DIMENSIONAL FLOW FIELDS USING THE COMPRESSIBLE REYNOLDS-AVERAGED NAVIER-STOKES EQUATIONS

Thomas H. Pulliam, Principal Investigator
Co-investigators: C. M. Hung, T. Barth, H. Bailey, R. Beam, D. Jespersen, M. Merriam, and Y. Liu
NASA Ames Research Center

Research Objective:
To implement and improve the various Navier-Stokes codes available in the CFD Branch at Ames on NAS's first high speed processor, the Cray-2.

Approach:
Port codes from CCF Cray-XMP to Cray-2 and debug the various versions of the compilers available on the Cray-2. Investigate the fundamental physics of two- and three-dimensional viscous flows using the increased memory capacity of the Cray-2.

Accomplishment Description:
Bailey and Beam: Implemented and applied their two-dimensional direct method Navier-Stokes code on the Cray-2. Analyzed Newton's method and approximations to Newton's method for Navier-Stokes.

Barth: Development of an implicit TVD scheme for both two- and three-dimensional Euler and Navier-Stokes equations. Initial use of RGL (remote graphics library) while investigating method accuracy and convergence. Use of RGL version of PLOT3D.

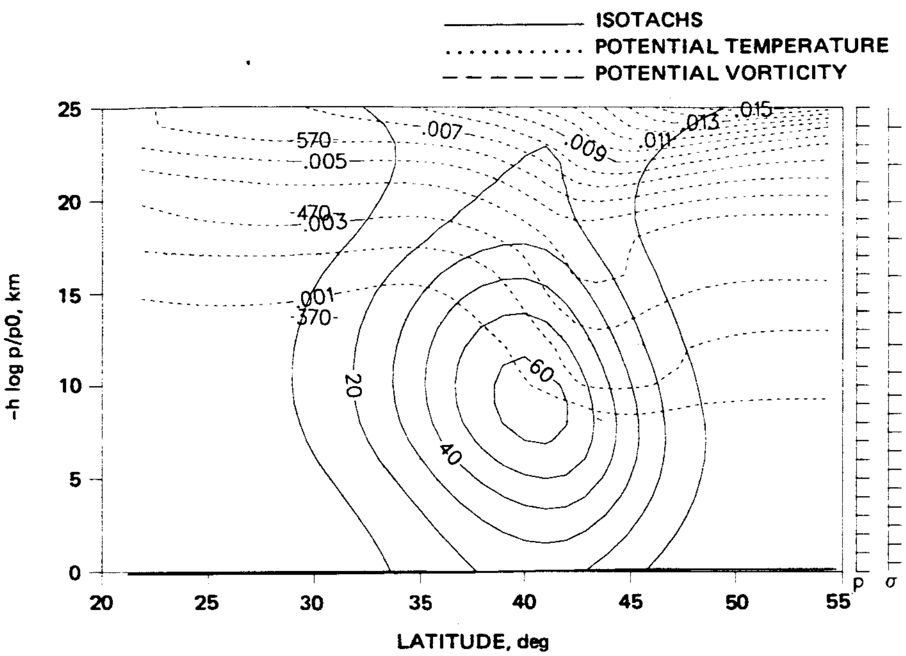
Pulliam: Ported ARC2D and ARC3D to the Cray-2. Applied codes to various flow problems such as circulation control airfoils, viscous airfoils, and three-dimensional simple bodies at high angle of attack. Investigated the cellular automata Navier-Stokes approach (jointly with Alan Wray). Investigated fractal geometry and dimension of Lorenz equations and a quadratic map using three-dimensional complex variables.

Hung: Modest use in trying to implement his three-dimensional Navier-Stokes Code.

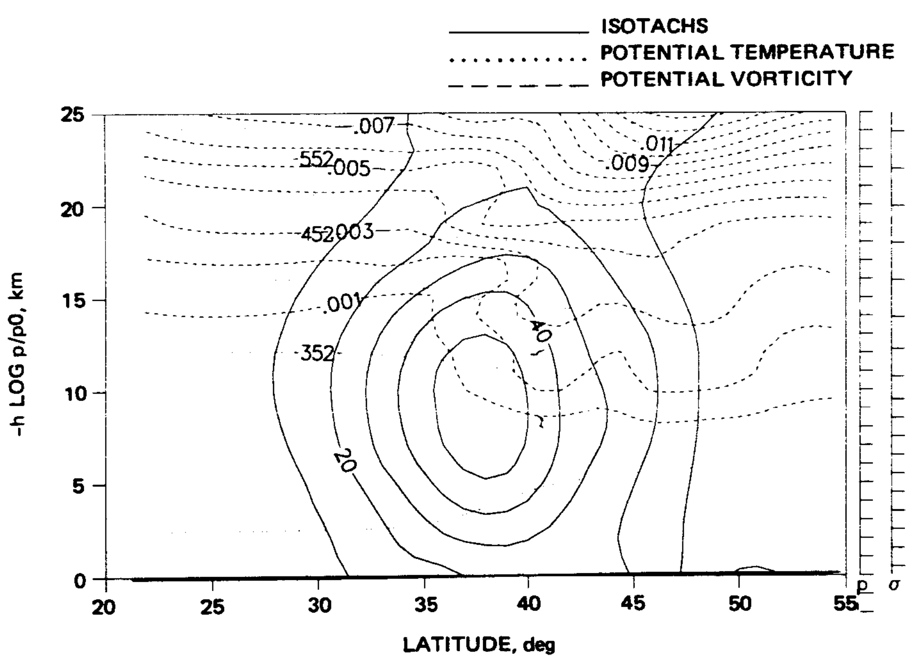
Merriam and Jespersen: Little use.

Significance:
Successful port of Cray-XMP codes to Cray-2. Improvement and further development of the codes ARC2D and ARC3D which are the kernels of many of the research codes used throughout the NAS community.

Cross Section, 126.29 deg. lon., 0 hr



Cross Section, 126.29 deg. lon., 108 hr



Height-latitude cross sections of wind speed (meters per second, solid), potential temperature (degrees Kelvin, dotted), and potential vorticity (dashed), for initial state (above) and after 108 hours of simulated time (right). Note the "folding" of potential vorticity surfaces in the 144 hour case at 35–40 degrees latitude and 10–15 km altitude.

PFISTER: UNDERSTANDING THE EFFECTS OF INITIAL CONDITIONS ON TROPOPAUSE FOLDS

TURBINE "HOT-STREAK" ANALYSIS

Man Mohan Rai, Principal Investigator
NASA Ames Research Center

Research Objective:
To determine the effect of combustor exit gas temperature inhomogeneities on turbine stages.

Approach:
To simulate the flow through a turbine rotor-stator configuration using a time-accurate Navier-Stokes code. The inlet conditions are adjusted to contain hot streaks of gas amidst the rest of the flow which is at design temperature. The subsequent interaction of the hot gases with the rotor blades is calculated and graphically animated.

Accomplishment Description:
The "hot-streak" problem was solved using a multizone Navier-Stokes rotor-stator code that was developed earlier. The mean surface temperatures and the temporal fluctuations of the temperature at the surfaces of the airfoils and in the regions surrounding the airfoils were calculated. The temperature distribution in the region of interest at a particular instant is shown in the attached photograph. The calculation showed that the surface temperature in certain areas of the rotor surface were much higher than the average temperature, thus indicating that cooling systems cannot be based on average temperatures. This information can be very important in redesigning the cooling systems for the rotor airfoils. This calculation took approximately 6 hours of computing on the Cray-2 and used 2 million words of core memory.

Future Plans:
The current calculation was two-dimensional in nature and, hence, did not yield hub to tip variations in temperature. A three-dimensional calculation will be performed in the near future with a code that is currently under development.

DEVELOPMENT OF A ZONAL APPROACH TO V/STOL FLOW FIELD ANALYSIS

Christopher L. Reed, Principal Investigator
General Dynamics

Research Objective:
Test and validate a propulsive-jet flow field analysis method which is based on the 3-D Navier-Stokes equations.

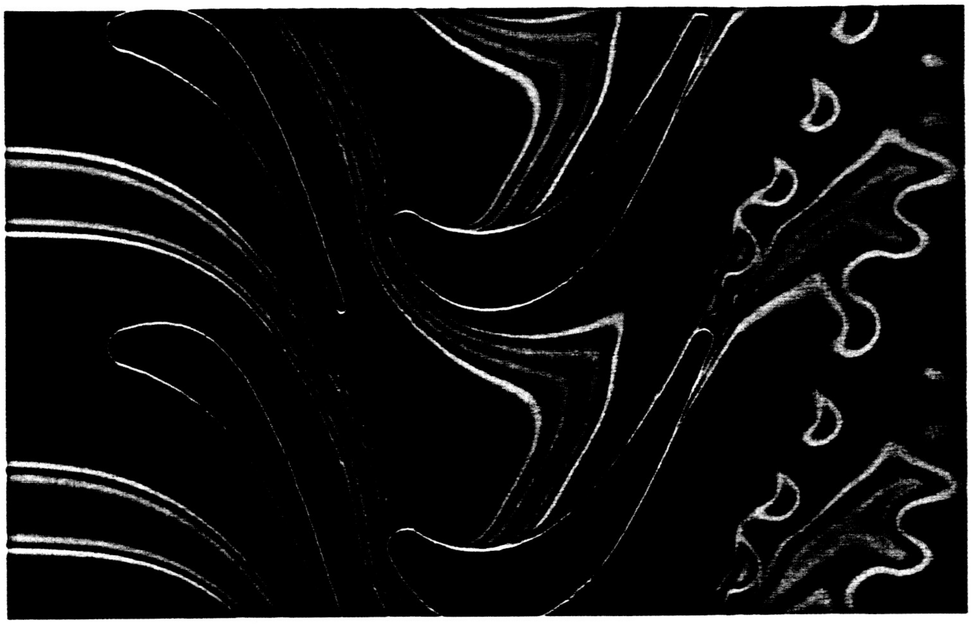
Approach:
Analyze the Mach 0.3 flow over a flat plate with a Mach 1.5 jet injected normal to the freestream and compare with experimental data.

Accomplishment Description:
One analysis was carried out and qualitative comparisons with test data made. The results of this study tend to point out areas of concern rather than provide definitive answers to the numerous problems associated with propulsive-jet flow field analysis. The computational results were not readily comparable to the experimental data because of significant differences between the two data sets.

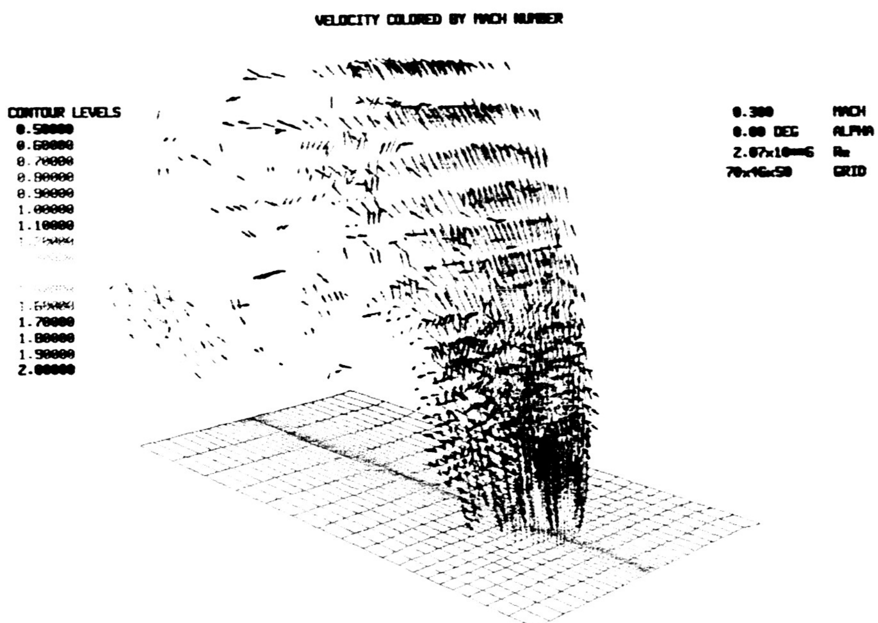
Significance:
Much can be learned from this study and applied to future efforts to use 3-D Navier-Stokes equations to analyze jets-in-crossflow.

Future Plans:
None (project not funded).

Publications:
Propulsive-Jet Flow Field Analysis Using the Three-Dimensional Navier-Stokes Equations. NASA Contractor Report, to be published.



RAI: TURBINE "HOT STREAK" ANALYSIS



REED: DEVELOPMENT OF A ZONAL APPROACH TO V/STOL FLOW FIELD ANALYSIS

THREE-DIMENSIONAL CAVITY

Donald P. Rizzetta, Principal Investigator
Air Force Wright Aeronautical Laboratories

Research Objective:
To numerically simulate the self-sustained oscillatory fluid behavior generated by a supersonic stream flowing over a three-dimensional cavity.

Approach:
The oscillatory cavity flow field was simulated numerically by time integration of the unsteady compressible three-dimensional Reynolds-averaged Navier-Stokes equations.

Accomplishment Description:
The self-sustained oscillatory motion within the cavity which is commonly observed experimentally has been reproduced numerically. Details of the flow field structure have been elucidated, and it is verified that the fundamental behavior of the unsteady phenomena is two-dimensional. Comparison with experimental data was made in terms of the mean static pressure and overall acoustic sound pressure levels within the cavity, as well as with the acoustic frequency spectra of the oscillation along the cavity floor and rear bulkhead. Total processing time required approximately 92.3 CPU hours and 4.8 megawords of storage capacity.

Significance:
The computation marks the first Navier-Stokes simulation of a three-dimensional cavity flow, and provides insight into the fluid behavior of this configuration. From a practical standpoint, the study is applicable to the flows in aircraft weapons bays and wheel wells.

Future Plans:
Prepare a dynamic display.

Publications:
Numerical Simulation of Supersonic Flow Over a Three-Dimensional Cavity. To be presented as AIAA Paper 87-1288 at the AIAA 19th Fluid Dynamics, Plasma Dynamics, and Lasers Conference, June 8-10, 1987. This paper has also been submitted for publication in the AIAA Journal.

3D VISCOUS FLOW IN HIGH-SPEED INLETS

William C. Rose and Edward W. Perkins, Principal Investigators
NASA Langley Research Center

Research Objective:
Compute inlet flow fields with 2D and 3D Navier-Stokes equations.

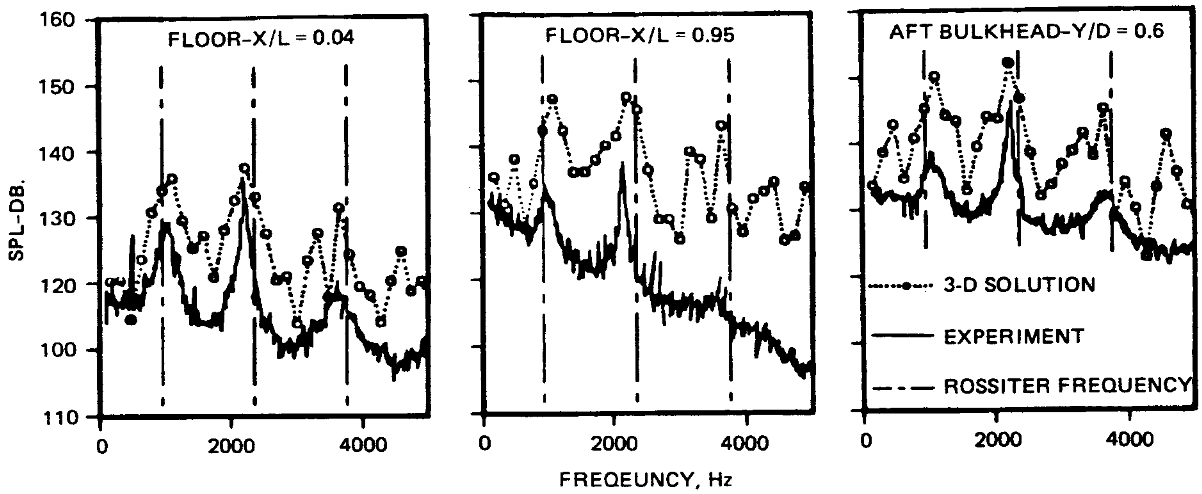
Approach:
Implement the existing NASA-Langley "Kumar" internal flow code (based on MacCormack's time-dependent algorithm) that has been run successfully on the XMP.

Accomplishment Description:
Approximately 20 hours of CPU time were expended with an average of 20 M words of run-time memory. Compiler problems were finally overcome and early stages of a solution of a Mach 5 engine inlet were obtained on a 201X61X51 grid. These early results are insufficient to show; however, they pointed to areas for improvement in speed and input/output problems.

Significance:
With the large run-time memory, solutions to this type of inlet flow field can be obtained without labor intensive partitioning of the grid.

Future Plans:
The modified code resulting from the IIOC is now (April 1987) operational on the NAS. The code is being used to solve 3D inlet and nozzle flow fields.

FREQUENCY SPECTRA



RIZZETTA: THREE-DIMENSIONAL CAVITY

HYPERSONIC JET INTERACTION

J. S. Shang, Principal Investigator
Co-investigator: D. L. McMaster
Air Force Wright Aeronautical Laboratories

Research Objective:
Develop basic knowledge and predictive methodology for a strong inviscid-viscous jet interaction phenomenon.

Approach:
Numerically simulate the jet interaction by solving the ensembled Navier-Stokes equations and validate the numerical solution by comparing with experimental data.

Accomplishment Description:
A side-by-side experimental and numerical simulation of a jet issuing normally from surfaces of sharp- and blunt-nosed ogive-cylinders in a hypersonic flow was accomplished for the first time. The experimental and numerical results were in reasonable accord. The separated flow field immediately adjacent to the jet exhibited a complex quadruple horseshoe vortical structure. The cross section of the computed downstream jet plume revealed a kidney shape which is identical to the classic experimental observation. A total of one hundred twenty CPU hours of single processed Cray-2 time was used for the three numerical simulations. The average memory was about five million 64-bit words.

Significance:
The numerical results represent the first ever solution of the Reynolds-averaged Navier-Stokes equations of a jet interacting phenomenon. After verification with experimental data, this numerical procedure will be used to analyze other problems of current interest, as described in the next section.

Future Plans:
Apply the developed procedure to the jet interaction to supersonic combustion and phenomenon associated with an aerospace vehicle which requires spinning motion for flight stability.

Publications:
AIAA Paper 87-0057, Interaction of Jet in Hypersonic Cross Stream.
AIAA Paper 87-1441, Supersonic Transverse Jet From a Rotating Ogive Cylinder Into a Hypersonic Flow (in preparation).

THREE-DIMENSIONAL FLOW SIMULATION

Joseph L. Steger, Principal Investigator
Co-investigators: Yehia Rizk, Lewis Schiff, and Susan Ying
NASA Ames Research Center

Research Objective:
To investigate and develop a general purpose three-dimensional flow solver for application to hypersonic, high-alpha, and powered-lift viscous flows.

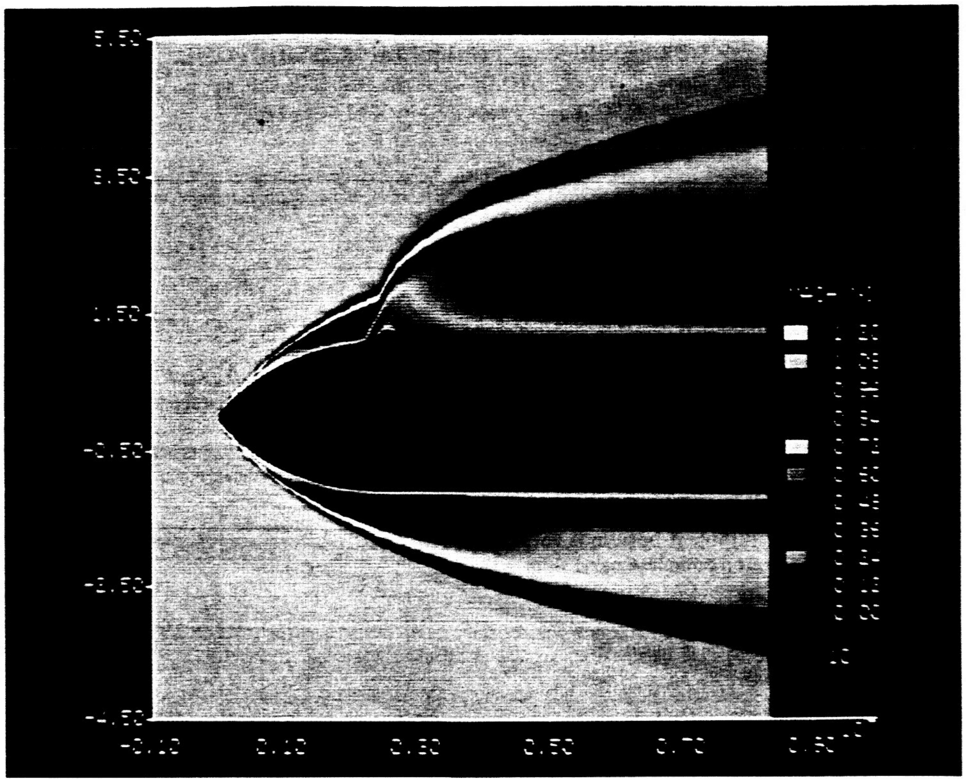
Approach:
To refine a partially flux-split, two-factor implicit finite difference scheme for the three-dimensional thin layer Navier-Stokes equations, F3D, and to test the code using high resolution 3D grids.

Accomplishment Description:
The F3D code was used in the early IIOC stages of the Cray-2 for computing the viscous flow about bodies at high alpha and NASP-like configurations. Three-dimensional grids that were much larger than we have previously exercised were used to resolve three-dimensional viscous separated flow phenomena.

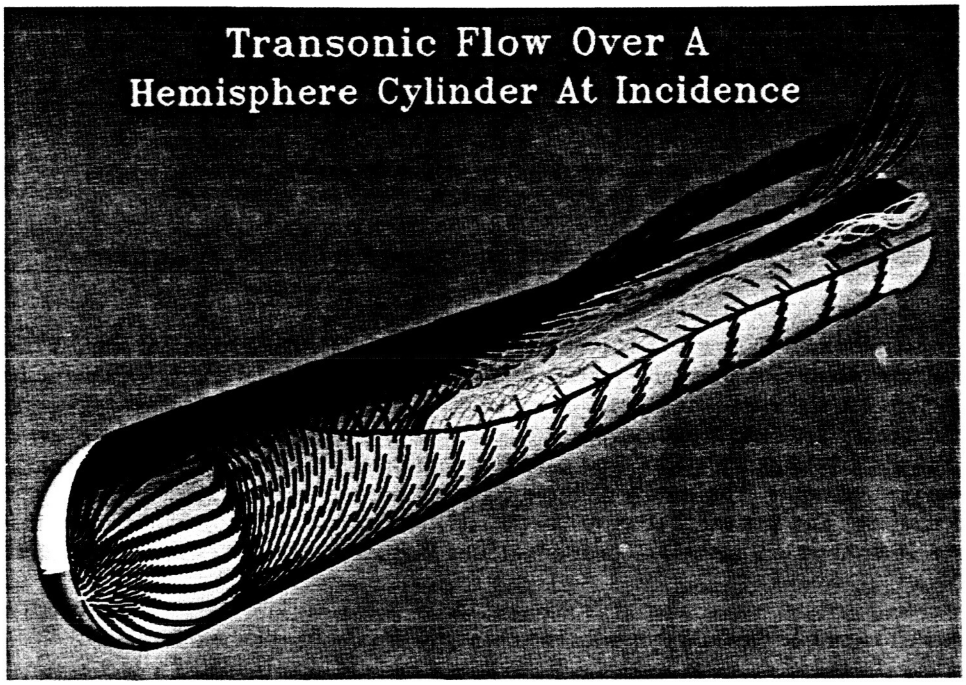
Significance:
NASP-like configurations were successfully simulated, and we found that for high-alpha flow it will be necessary to adequately resolve, and not model, the outer vortical flow features.

Future Plans:
To continue to refine the code and algorithm to understand and predict complex viscous flow phenomena about practical configurations.

Publications:
Ying, S. X., Steger, J. L., Schiff, L. B., and Baganoff, D.: Numerical Simulation of Unsteady Viscous, High Angle of Attack Flows Using a Partially Flux-Split Algorithm. AIAA Paper 86-2179.
Rizk, Y., Chaussee, D., and Steger, J. L.: Numerical Simulation of the Hypersonic Flow Around Lifting Vehicles. AGARD, 1987.



SHANG: HYPERSONIC JET INTERACTION



STEGER: THREE DIMENSIONAL FLOW SIMULATION

CFD FOR NAVAL APPLICATIONS

Chao-Ho Sung, Principal Investigator
Co-investigators: Cheng-Wen Lin and Chu Cheng
David W. Taylor Naval Ship R&D Center

Research Objective:
The development of the capability to predict the flow fields about naval ships in order to gain the understanding of the physics of the flow field and the application of this understanding to aid the innovative design of naval ships.

Approach:
The 3D incompressible Reynolds-averaged Navier-Stokes equations with an appropriate turbulence model are solved by the finite-difference approach.

Accomplishment Description:
A computer code based on the finite-volume formulation and an explicit one-step multistage Runge-Kutta time scheme has been developed. Improvements in computational efficiency have been demonstrated in several areas including a four-stage Runge-Kutta scheme, the estimate of a local time step size with the viscous effect included and the implementation of the boundary conditions at the farfield, the solid wall and the symmetric plane. The prediction of the turbulent horseshoe vortex flow past an airfoil/flat-plate juncture indicates good agreement with the experimental data. An average run in a grid of 65X49X45 at 1500 time steps used about 40 minutes of Cray-2 time and about 5 MW of memory.

Significance:
The ability to predict the incompressible turbulent horseshoe vortex flow past an airfoil/flat-plate juncture is a major step forward in developing the capability to predict the flow field about a complete naval ship.

Future Plans:
Research will be continued in improving both the accuracy and efficiency of numerical techniques. Particular emphasis will be in the higher accuracy of the discretization scheme, artificial dissipation model, multizone method, multigrid method and parallel processing. Practical applications to the design of appendages will also be pursued.

Publications:
1. An Explicit Runge-Kutta Method for 3D Turbulent Incompressible Flows. Submitted to the 7th GAMM Conference on Numerical Methods in Fluid Mechanics, September 1987.
2. Numerical Proof of A. D. Young's Conjecture on a Structure in the Wake of a Wing/Body Juncture. Presented at SNAME H-11 Panel Meeting at NASA/Langley, April 6, 1987, manuscript in preparation to be submitted to Journal of Fluid Mechanics.

NUMERICAL SOLUTION OF CHEMICAL NONEQUILIBRIUM FLOW FIELD AROUND THE SPACE SHUTTLE ORBITER

John C. Tannehill, Principal Investigator
Co-investigator: Dinesh K. Prabhu
Iowa State University

Research Objective:
Develop a computer code to compute the viscous, reacting flow around hypersonic vehicles.

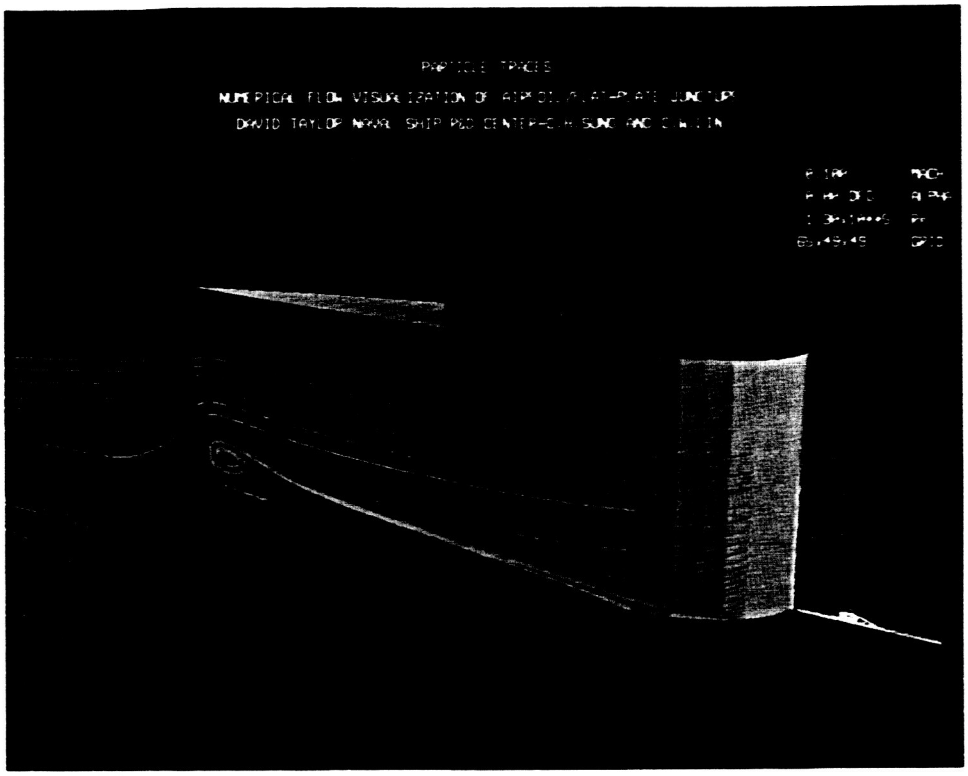
Approach:
The reacting parabolized Navier-Stokes equations are solved using an implicit finite-difference scheme.

Accomplishment Description:
A new parabolized Navier-Stokes (PNS) code has been developed to compute the hypersonic laminar flow of a multicomponent, chemically reacting mixture of thermally perfect gases over two-dimensional and axisymmetric bodies. The new PNS code solves the gas dynamic and species conservation equations in a coupled manner using a noniterative, implicit, space-marching finite-difference method. The conditions for well-posedness of the space-marching method have been derived from an eigenvalue analysis of the governing equations. The code has been used to compute hypersonic laminar flow of chemically reacting air over wedges and cones. The results of these computations are in good agreement with the results of reacting boundary-layer calculations. The present 2-D/axisymmetric calculations required approximately 120 sec of CPU time on the Cray-2 computer. It is estimated that 3-D calculations around a typical hypersonic vehicle will require 2-3 hr of CPU time.

Significance:
This PNS code is the first to couple the fluid dynamics and chemistry in a noniterative manner.

Future Plans:
The code is currently being extended to three dimensions.

Publications:
Prabhu, D. K., Tannehill, J. C., and Marvin, J. G.: A New PNS Code for Nonequilibrium Flows. AIAA Paper 87-0284, January 1987.



SUNG: CFD FOR NAVAL APPLICATIONS

THREE-DIMENSIONAL NAVIER-STOKES ALGORITHM DEVELOPMENT

James L. Thomas, Principal Investigator
Co-investigators: Christopher L. Rumsey and W. Kyle Anderson
NASA Langley Research Center

Research Objective:
To develop efficient methods for large-scale computations based on solutions of the Reynolds-averaged Navier-Stokes equations.

Approach:
Develop and validate an implicit multigrid method based on an upwind differencing approach to the convective and pressure terms.

Accomplishment Description:
Solutions to two models at high angle of attack, a 75° swept delta wing and a prolate spheroid, have been obtained to date. Comparisons with detailed experimental data sets are ongoing and indicate generally good agreement. A converged solution with approximately 200,000 grid points requires 2-3 hours and 8 MW on the Cray-2 computer.

Significance:
Improved methodology (AIAA 87-1104 CP) incorporated and demonstrated for laminar and turbulent 3-D flows.

Future Plans:
Incorporation of multiple/imbedded-block approach for increased efficiency and generality. Application to F-18 geometry.

RUNGE-KUTTA SOLUTIONS FOR THE FLUID DYNAMIC EQUATIONS

Eli Turkel, Principal Investigator
ICASE/NASA Langley Research Center

Research Objective:
To find more efficient and accurate solutions to the Euler and Navier-Stokes equations.

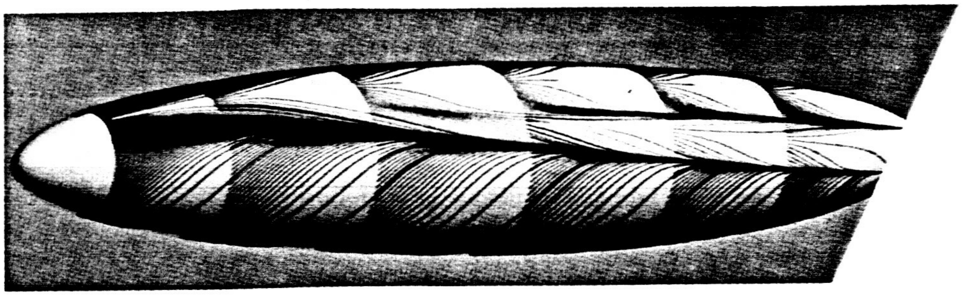
Approach:
We use a multistage Runge-Kutta algorithm accelerated with residual smoothing and multigrid.

Accomplishment Description:
The artificial viscosity used in the algorithm has been analyzed in detail. It was found useful to rewrite the fourth difference as a second difference of a second difference. In addition, the scaling should be different in each direction to allow for highly stretched grids.

Significance:
The multigrid solution to the Euler equation has been speeded up by 50%. More accurate solutions for viscous flow was also obtained.

Future Plans:
To extend these results to turbulent flow and also to three dimensions.

Publications:
Artificial Dissipation and Central Difference Schemes for the Euler and Navier-Stokes Equations. AIAA CFD Conference, Hawaii, June 1987.



THOMAS: THREE DIMENSIONAL NAVIER-STOKES ALGORITHM DEVELOPMENT

NAVIER-STOKES ANALYSIS OF 3-D CONFIGURATIONS

Dr. J. Vadyak, Principal Investigator
Co-investigators: D. M. Schuster, M. J. Smith, G. D. Shrewsbury, and R. Weed
Lockheed-Georgia Co.

Research Objective:
Develop Navier-Stokes methodology for prediction of flow fields over arbitrary aircraft configurations.

Approach:
Continue the development and refinement of viscous implicit 3-D finite-difference numerical techniques for both perfect and real gas simulations.

Accomplishment Description:
A generalized 3-D Navier-Stokes simulation algorithm has been developed which is applicable to a variety of aircraft components or to integrated aircraft configurations using zonal grid generation techniques. Versions of the algorithm exist for perfect gas, calorically imperfect gas, or a mixture of gases accounting for finite-rate thermo-chemical effects. An example of a typical simulation is shown in the figure which illustrates the surface pressure distribution over a generic hypersonic cruise vehicle at $M_\infty = 6.0$ and zero incidence. These results are for a full 3-D turbulent Navier-Stokes simulation using a calorically imperfect gas model. In the figure, high pressure is denoted by red, and low pressure is denoted by blue.

Significance:
Viscous turbulent 3-D flow field simulation for realistic vehicles of current interest have been obtained.

Future Plans:
Continue the development/refinement of these numerical techniques especially for simulations including finite-rate chemical kinetic effects.

HYBRID/FORTIFIED NAVIER-STOKES

William R. Van Dalsem, Principal Investigator
Co-investigator: Joseph L. Steger
NASA Ames Research Center

Research Objective:
Develop and apply the Hybrid/Fortified Navier-Stokes (FNS) scheme to projects of national importance.

Approach:
The FNS scheme is a method for coupling arbitrary formulations, algorithms, and grid topologies in an effort to obtain the most cost effective CFD simulation.

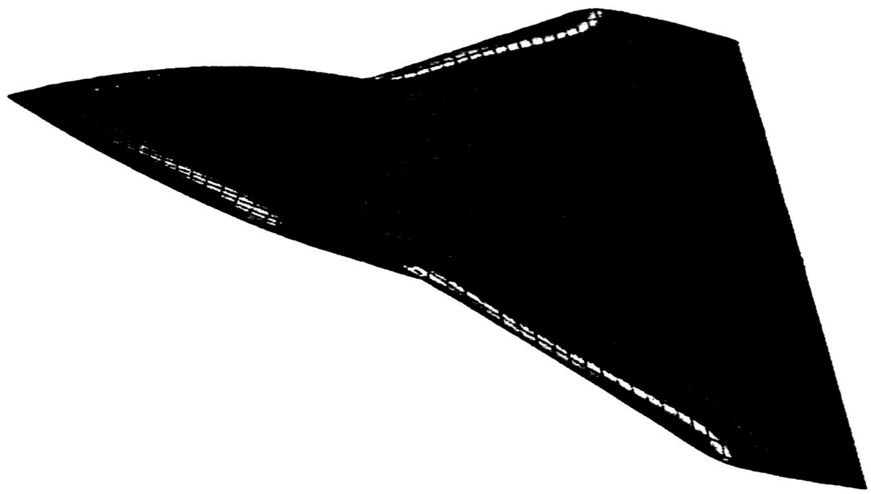
Accomplishment Description:
(~40 hours, 15 Mw of Cray-2 resources used.) Using the FNS approach, an unsteady three-dimensional boundary-layer algorithm, and the F3D Navier-Stokes code, accurate drag predictions for the separated flow over a finite aspect ratio wing where obtained with an order-of-magnitude savings of computer time over a standard Navier-Stokes scheme.

The jet in ground effect with a cross wind flow field (of primary interest in the design of the next generation of VSTOL aircraft) was studied extensively using the F3D/FNS code. The FNS scheme capability to apply internal boundary conditions was of great use in this simulation.

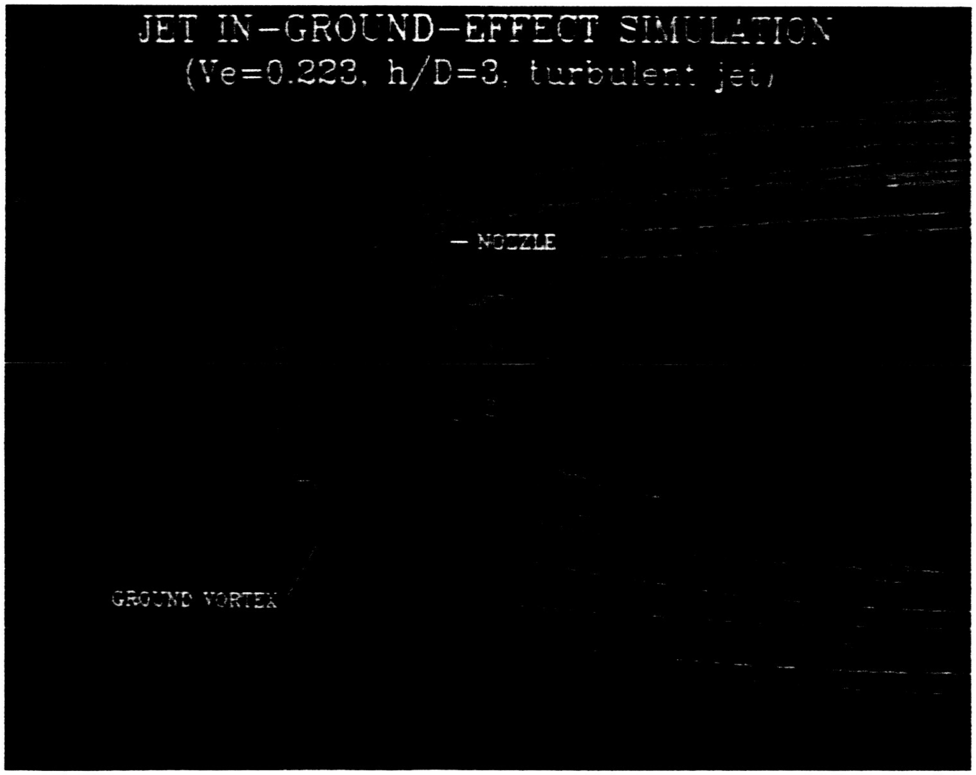
Significance:
The FNS scheme can be of use in accelerating most applications of Navier-Stokes equations. The VSTOL simulations add to the understanding of flows critical to the success of a number of national and international projects leading to the production of the next generation of high performance transport and fighter aircraft.

Future Plans:
Continue to explore the potentials of the FNS method in applications in the VSTOL area.

Publications:
The Fortified Navier-Stokes Approach. Workshop on Computational Fluid Dynamics, University of California, Davis.
Numerical Investigation of a Jet in Ground Effect Using the Fortified Navier-Stokes Scheme. Ground-Vortex Workshop, NASA Ames Research Center.



VADYAK: NAVIER-STOKES ANALYSIS OF 3-D CONFIGURATIONS



VAN DALSEM: HYBRID/FORTIFIED NAVIER-STOKES

HIGH REYNOLDS NUMBER, TRANSONIC VISCOUS FLOW OVER AIRCRAFT COMPONENTS

V. N. Vatsa, Principal Investigator
Co-investigators: N. Duane Melson and Ernst von Lavante
NASA Langley Research Center/Old Dominion University

Research Objective:
To study the multigrid acceleration of the iterative solution of the two-dimensional isenthalpic Euler equations.

Approach:
The assumption of constant total enthalpy was used to replace the Euler energy equation with an algebraic expression. The reduced set of three pde's was formulated using a flux-vector splitting method with MUSCL differencing. The resulting set of equations was solved using a two-factor approximate factorization scheme. Iterative convergence was accelerated using Full Approximation Storage (FAS) multigrid as well as Full Multigrid (FMG).

Accomplishment Description:
The iterative solution of the isenthalpic form of the two-dimensional Euler equations was successfully accelerated using multigrid. Using FAS and FMG, the lift coefficient (C_ℓ) of an NACA 0012 airfoil at a free-stream Mach number (M_∞) of 0.8 and an angle of attack (α) of 1.25° was predicted to within 1% of the fully converged value in the equivalent of only 30 iterations. Results predicted by the isenthalpic program were found to be in good agreement with results obtained by other researchers using significantly finer grids.

Significance:
The fast convergence obtained using FAS and FMG for the solution of the two-dimensional isenthalpic form of the Euler equations makes the solution of the Euler equations a useful engineering tool.

Future:
Viscous terms will be added to the previously described program so that solutions of the thin-layer Navier-Stokes (TLNS) equations may be obtained using the program. Research will involve seeking the same convergence rates for the TLNS solutions as for the Euler solutions. Work will also be performed on extensions to three dimensions.

Publications:
Melson, N. Duane, and von Lavante, Ernst: Multigrid Acceleration of the Isenthalpic Form of the Compressible Flow Equations. Presented at the Third Copper Mountain Conference on Multigrid Methods, April 6-10, 1987.

ALGORITHM DEVELOPMENT FOR THE NSC

Thomas A. Zang, Principal Investigator
Co-investigators: S. E. Krist and M. G. Macaraeg
NASA Langley Research Center

Research Objective:
To develop and validate algorithms suitable for transition and turbulence simulations on the Navier-Stokes Computer (NSC) which is being developed at Princeton University under NASA support.

Approach:
Develop algorithms suitable for local memory, communication-bound parallel processing computers, implement these algorithms on a conventional supercomputer (the Cray-2), verify their accuracy, and predict their performance on the NSC.

Accomplishment Description:
A simple finite-difference algorithm was developed for the simulation of isotropic turbulence. The most time-consuming portion of this algorithm (solution of Poisson and Helmholtz equations in three dimensions) was identified. A variety of classical and modern iterative schemes for solving these equations was examined. These were implemented on the Cray-2 and their performance was compared with the theoretical predictions. This information was used to project the performance of the NSC on the massive problems which it should be able to accommodate.

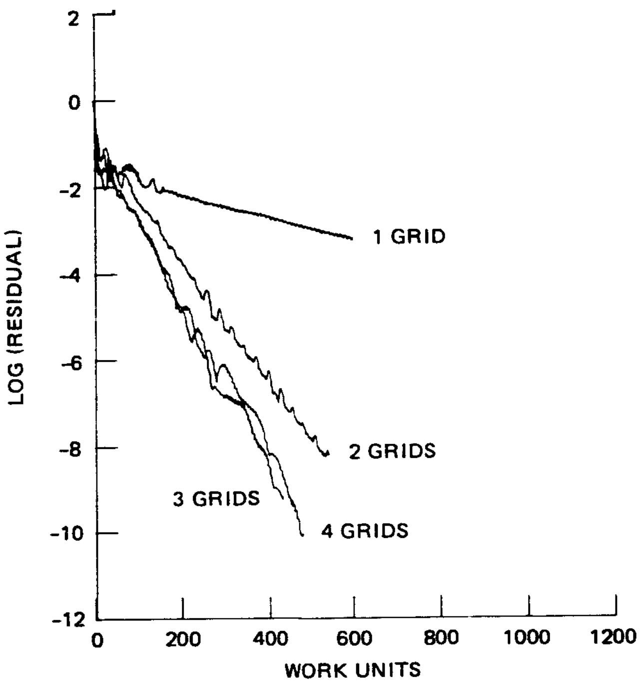
In addition, a different class of algorithms suitable for local memory parallel processors was examined. These are based on using spectral rather than finite-difference methods on subdomains.

A total of 70 CPU hours was used during IIOC.

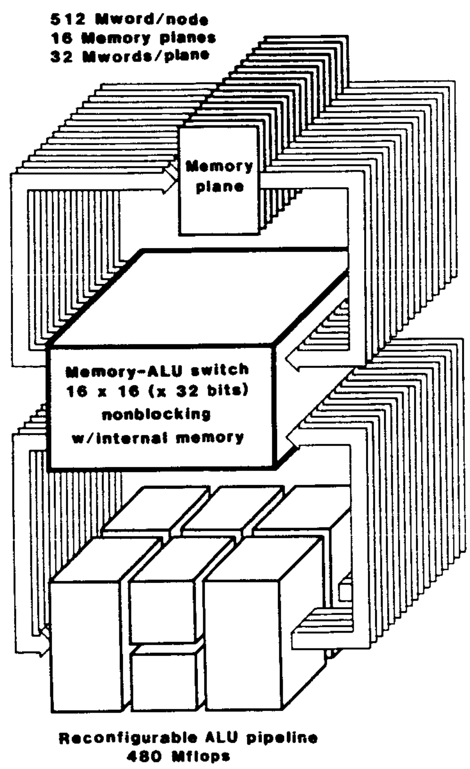
Significance:
The algorithm under development on the Cray-2 will be implemented on the NSC.

Future Plans:
Develop and verify algorithms for more complicated problems such as wall-bounded transition.

MULTIGRID ACCELERATION AT $M_\infty = 0.80$ AND $\alpha = 1.25°$



VATSA: HIGH REYNOLDS NUMBER, TRANSONIC VISCOUS FLOW OVER AIRCRAFT COMPONENTS



ZANG: ALGORITHM DEVELOPMENT FOR THE NSC

DIRECT AND LARGE-EDDY SIMULATION OF COMPRESSIBLE TURBULENCE

Thomas A. Zang, Principal Investigator
Co-investigators: G. Erlebacher and N. MacGiolla Mhurris
NASA Langley Research Center

Research Objective:
To develop a subgrid-scale model for compressible, turbulent flows. Such a model would permit the simulation of higher Reynolds number flows than is presently possible.

Approach:
Perform direct numerical simulation of turbulent compressible flows at low Reynolds number and compare the exact turbulent fields with those predicted by the model. Next, perform large-eddy simulation of higher Reynolds number flows using the new subgrid-scale model.

Accomplishment Description:
A spectral collocation algorithm was developed for performing direct and large-eddy simulations of homogeneous, compressible, turbulent flows. The algorithm includes physically realistic molecular transport properties and also operates efficiently in the limit of very low Mach number flows.

An extensive data base has been generated over the relevant Mach number range for isotropic turbulent flows. The calculations were performed on 96^3 and 128^3 grids (using 15 Mwords and 35 Mwords, respectively). These data have been used to validate and calibrate a subgrid-scale model for compressible, homogeneous turbulent flows. In the limit of zero Mach number, the model reproduces earlier results for incompressible turbulence, and the model constants for the new, compressible version are relatively insensitive to Mach number.

A total of 250 CPU hours were used during IIOC. (Twice as many CPU hours were used during the pilot usage period.)

Significance:
This work was the first such subgrid-scale modeling effort for compressible flow. It represents an essential first step in extending the large-eddy simulation technique to aerodynamic flows.

Future Plans:
Study other types of homogeneous flows such as those involving uniform shear and develop and verify subgrid-scale models for inhomogeneous, wall-bounded compressible turbulent flows.

Publications:
Zang, T. A., Drummond, J. P., Erlebacher, G., Speziale, C., and Hussaini, M. Y.: Numerical Simulation of Transition, Compressible Turbulence and Reacting Flows. AIAA Paper 87-0130, 1987.
Erlebacher, G., Hussaini, M. Y., Speziale, C. G., and Zang, T. A.: Toward the Large-Eddy Simulation of Compressible Turbulent Flows. ICASE Report No. 87-20, 1987.

TRANSITION TO TURBULENCE

Thomas A. Zang, Principal Investigator
NASA Langley Research Center

Research Objective:
To study the process by which fluid motions transition from a smooth, laminar state to a chaotic, turbulent state. At present, only the early stages of this transition process are well understood.

Approach:
Perform numerical simulation of simple, low-speed channel and boundary-layer flows under controlled conditions in the transitional regime. The simulations are performed for the time-dependent, incompressible Navier-Stokes equations using highly accurate spectral methods.

Accomplishment Description:
High resolution channel flow simulations (using up to 30 Mwords) have disclosed the existence and clarified the structure of finite amplitude center mode instabilities. These are an alternative path to turbulence which had heretofore received no attention.

A parametric study was performed of nonlinear effects upon laminar flow control techniques in water boundary layers (using up to 2 Mwords). Temperature fluctuations were shown to have a destabilizing effect in the nonlinear regime (they are stabilizing in the linear regime), predominantly due to two-dimensional effects.

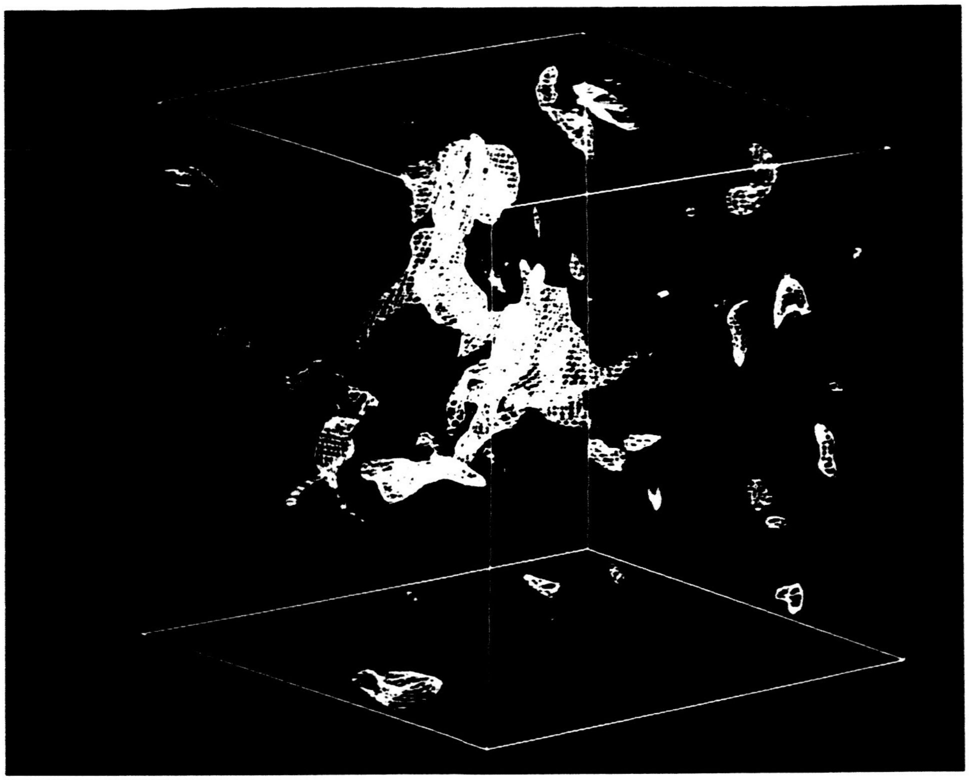
Several high resolution simulations of the latter stages of boundary-layer transition were conducted (using up to 65 Mwords). A large data base has been generated. This is presently being analyzed for clues to the nature of tertiary instabilities in boundary layers.

A total of 400 CPU hours were used during IIOC. (A comparable amount was also used during the pilot usage period.)

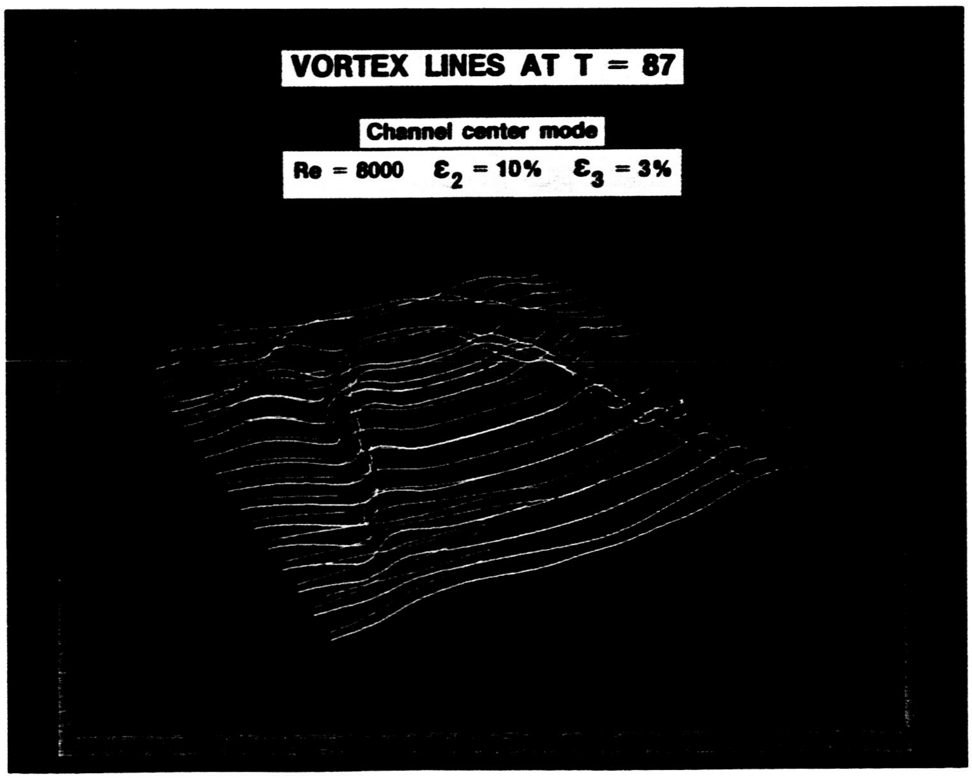
Significance:
These calculations have disclosed the nature of some of the processes by which laminar flows become turbulent. They also form the basis for future work on transition in high-speed compressible flows.

Future Plans:
Continue the data analysis and graphical representation of the tertiary instability.

Publications:
Zang, T. A., Drummond, J. P., Erlebacher, G., Speziale, C., and Hussaini, M. Y.: Numerical Simulation of Transition, Compressible Turbulence and Reacting Flows. AIAA Paper 87-0130, 1987.
Bushnell, D. M., Hussaini, M. Y., and Zang, T. A.: Sensitivity of LFC Techniques in the Nonlinear Regime. Symposium on Natural Laminar Flow and Laminar Flow Control Research. NASA CP, 1987.



ZANG: DIRECT AND LARGE-EDDY SIMULATION OF COMPRESSIBLE TURBULENCE



ZANG: TRANSITION TO TURBULENCE

INDEX OF RESEARCH SITES

PAGE 58 INTENTIONALLY BLANK

GPO 789-187/79006